

AD-A169 471

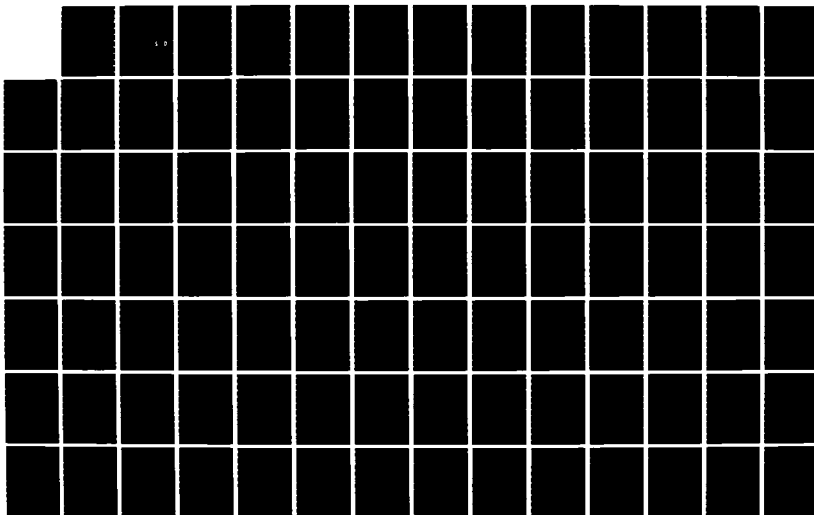
ADAPTIVE GRID TECHNIQUES FOR ELLIPTIC FLUID-FLOW
PROBLEMS(U) STANFORD UNIV CA CENTER FOR LARGE SCALE
SCIENTIFIC COMPUTATION S C CARUSO ET AL. DEC 85
CLASSIC-85-10 N00014-82-K-0335

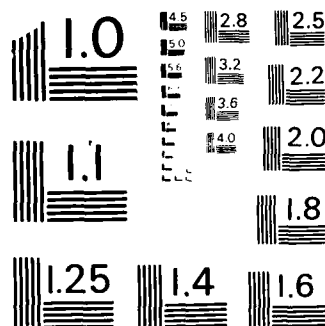
1/3

UNCLASSIFIED

F/G 20/4

NL





MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS - 1963 - A

8

CLaSSiC Project
Manuscript CLaSSiC-85-10

December 1985

Adaptive Grid Techniques for Elliptic Fluid-Flow Problems

AD-A169 471

S. C. Caruso

J. H. Ferziger

J. Oliger

DTIC
ELECTE
JUN 26 1986
S D

DTIC FILE COPY

Center for Large Scale Scientific Computation
Building 460, Room 313
Stanford University
Stanford, California 94305



DISTRIBUTION STATEMENT A

Approved for public release
Distribution Unlimited

ADAPTIVE GRID TECHNIQUES FOR ELLIPTIC FLUID-FLOW PROBLEMS

by

S. C. Caruso, J. H. Ferziger and J. Oliger

Work done within the
Center for Large Scale Scientific Computing
(CLaSSiC)

Under support from
The Office of Naval Research
Grant N00014-82-K-0335

December 1985

Acknowledgments

The authors thank Dr. Marsha Berger for providing assistance in understanding her data structures and grid-generation subroutines. The authors also thank Professor Abdel Zebib for supplying us with his working version of the SIMPLER code.

The financial support for this project was provided by the Office of Naval Research through the Center for Large-Scale Scientific Computation under contract N00014-82-K-0335. It is greatly appreciated.

Mrs. Ruth Korb has done an excellent job of editing and typing this report. Her efforts are very much appreciated.

Accession For	
NTIS CRA&I	<input checked="checked" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By <i>per ltr.</i>	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	



Abstract

We describe adaptive grid techniques for elliptic fluid-flow problems. The primary applications are to flows described by the steady, incompressible laminar and Reynolds-averaged Navier-Stokes equations. The method can be extended to other elliptic flows. The current version of the adaptive method is designed for use on a single, uniform, rectangular grid or unions of such grids. Other geometries could be handled by means of commonly used mapping or composite grid techniques.

Our method is an extension of a local refinement technique developed by Berger for systems of hyperbolic equations. Local refined grids are overlaid on a coarser base grid. Recursive use of this technique allows an arbitrary degree of grid refinement. In two dimensions, the refined grid consists of uniform rectangles having arbitrary rotation. Regions needing refinement are defined using local error estimates.

The base grid remains fixed, and refinements are added as needed. This method contrasts with global refinement methods, in which a fixed number of points on a single grid are iteratively redistributed according to some criterion such as the local solution gradient, curvature, etc.

Two classes of elliptic flows are identified; they are characterized as having strong or weak viscous-inviscid interactions. Adaptive solution strategies, active and passive, respectively, are developed for each class.

The passive method is applied to linear problems in one and two dimensions. In 2-D, the refined grids automatically align with the flow, thereby minimizing numerical diffusion. The adaptive method is shown to be more efficient than using a uniform fine grid.

The SIMPLER method is used to solve the steady, laminar, incompressible Navier-Stokes equations. Central differencing of the convective terms is implemented with the defect-correction method to stabilize the solution method for all cell Reynolds numbers. Smooth solutions are calculated for cell Reynolds numbers as high as 150, indicating that the commonly used restriction, $Re_{\Delta x} < 2$ is too severe. Uniform-grid

calculations are performed for the laminar backstep flow. Patankar's power-law scheme is shown to be less accurate than central differencing, and only first-order accurate.

Richardson-estimated solution and truncation errors are compared to accurate estimates of the same quantities for the backstep flow. The solution error is well predicted. The truncation-error estimates are less accurate, but they reliably indicate where grid refinement is required.

Active-adaptive calculations of the backstep are made, using boundary-aligned refinement. At $Re = 100$, the adaptive calculation has comparable accuracy, but is six times faster than a uniform-grid calculation; the advantage is greater at higher Reynolds numbers. Adaptive calculations are also made at higher Reynolds numbers. The calculations agree well with the experimental data and other calculations.

Table of Contents

	Page
Acknowledgments	iii
Abstract	iv
List of Figures	viii
List of Tables	xi
Nomenclature	xii
Chapter	
1. INTRODUCTION	1
1.1. Motivation	1
1.2. Adaptive Grid Methods	2
1.2.1. Global Refinement Methods	3
1.2.2. Local Refinement Methods	10
2. REVIEW OF BERGER'S METHOD	17
2.1. Overview	17
2.2. Grid Description	17
2.3. Adaptive Solution Procedure	19
2.4. Error Estimation	21
2.5. Refined-Grid Generation	23
2.6. Boundary Values for Refined Grids	24
2.7. Data Structures	26
3. STRATEGY FOR ELLIPTIC EQUATIONS	31
3.1. Overview	31
3.2. Influence of Elliptic Equations on Adaptive Approach	31
3.3. Two Adaptive Strategies	32
3.4. Passive Method	35
3.4.1. Strategy	35
3.4.2. Summary of Theoretical Results	36
3.5. Active Method	38
3.6. Solution on Overlapping Grids	42
3.7. Error Estimation	45
3.8. Initial Guesses and Boundary Values for Refined Grids	49
4. APPLICATION TO ONE-DIMENSIONAL BOUNDARY-VALUE PROBLEMS	57
4.1. Introduction	57
4.2. Numerical Method	58
4.3. Adaptive Procedure	58
4.4. Numerical Results	59
4.5. Conclusions	63
5. APPLICATION TO A TWO-DIMENSIONAL, LINEAR, CONVECTION- DIFFUSION PROBLEM	75
5.1. Introduction	75
5.2. Numerical Method	76
5.3. Adaptive Procedure	77
5.4. Numerical Results	79
5.5. Conclusions	82
6. NUMERICAL SOLUTION OF THE NAVIER-STOKES EQUATIONS	87
6.1. Navier-Stokes Equations	87
6.1.1. The Incompressible Navier-Stokes Equations	87

6.1.2.	Reynolds-Averaged Equations	88
6.1.3.	Eddy-Viscosity Models	89
6.1.4.	Steady, 2-D Laminar Equations	91
6.2.	Staggered Grid	92
6.3.	Finite Differences	93
6.3.1.	Pressure Difference	94
6.3.2.	Diffusion Terms	94
6.3.3.	Convective Terms	96
	a. Central Differencing	97
	b. Upwind Differencing	98
	c. Hybrid Differencing	99
	d. QUICK Differencing	100
6.4.	SIMPLER Solution Technique	101
6.4.1.	Pressure Calculation	102
6.4.2.	Momentum Calculation	103
6.4.3.	Velocity and Pressure Corrections	104
6.4.4.	Solution of Linear Systems	105
6.5.	Implementation of Central Differencing for $Re_{\Delta x} > 2$	106
6.6.	Numerical Conservation	108
7.	ADAPTIVE NAVIER-STOKES SOLVER	115
7.1.	Summary of Adaptive Process	115
7.2.	Implementation of the Active Solution Method	116
7.3.	Treatment of Boundary Conditions	118
7.3.1.	Interpolation of Fine-Grid Boundary Con- ditions	118
7.3.2.	Modification of Coarse-Grid Boundary Conditions	119
7.3.3.	Interpolation and Conservation for Rotated Grids	120
7.4.	Error Estimation	122
8.	APPLICATION TO THE LAMINAR BACKSTEP	129
8.1.	Description of the Problem	129
8.1.1.	Experiment of Arnaly et al.	129
8.1.2.	Computational Model	130
8.2.	Uniform Grid Calculations	131
8.2.1.	Velocity Profiles	132
8.2.2.	Comparison of Convective Difference Schemes	133
8.2.3.	Exact vs. Estimated Errors	135
8.3.	Preliminary Adaptive Calculations at $Re = 100$	137
8.3.1.	Notation for Refined Grids	138
8.3.2.	Convergence of the Active Solution Method	138
8.3.3.	Iteration Strategy: Inner vs. Outer	139
8.3.4.	Effect of Internal Fine-Grid Boundary Location	140
8.4.	Adaptive Performance Evaluation for $Re = 100$	141
8.5.	Adaptive Results for $100 < Re < 600$	142
9.	CONCLUSIONS AND RECOMMENDATIONS	179
	REFERENCES	183
Appendix A	QUANTIFICATION OF THE STRENGTH OF THE VISCOUS- INVISCID INTERACTION	189

List of Figures

Figure	Page
2.1. Overlapping, component grids	28
2.2. Example of overlaid grid structure	28
2.3. Time integration on three-level grid structure	29
2.4. Richardson extrapolation for time-explicit method	29
2.5. Tree data structure	30
3.1. Developing boundary layer in a plane channel	52
3.2. Quantification of boundary-layer coupling	52
3.3. Example of newly refined regions	53
3.4. Notation for active solution on two-level grid system	53
3.5. Difference stencils and coarse-grid correction terms	54
3.6. Notation for solution on overlapping grids	54
3.7. Local cell coordinates for bilinear interpolation	55
4.1. Example 1-D grid structures	6 ⁵
4.2(a). Simple boundary layer: base grid solution.	66
4.2(b). Simple boundary layer: solution on second-level grid	67
4.2(c). Simple boundary layer: solution on third-level grid	68
4.2(d). Simple boundary layer: complete adapted solution	69
4.2(e). Simple boundary layer: central difference solution on base grid	70
4.3. Adapted solution for two-boundary-layer problem	71
4.4. Adapted solution for internal boundary-layer problem	72
4.5. Adapted solution for boundary layer with non-constant outer solution	73
4.6. Adapted solution for problem with a turning point and a boundary layer	74
5.1. Schematic of linear convection-diffusion problem	83

5.2(a).	Initial refinement region	84
5.2(b).	First-level refined grids	84
5.2(c).	First-level refinement region	85
5.2(d).	Resulting refined grids	85
5.3.	Adaptive vs. uniform grid performance	86
6.1.	Staggered grid geometry	111
6.2.	Centered differences on staggered and nonstaggered grids . .	112
6.3.	Finite volume for u	113
6.4.	Difference stencil for boundary cells	113
6.5.	Finite volume for p	114
6.6.	Line-by-line solution method	114
7.1.	Boundary-aligned refined-grid geometry	124
7.2.	Notation for active solution on two-level, 2-D grid system .	125
7.3.	Conservative coarse-to-fine grid interpolation	126
7.4.	Interpolation for rotated grids	127
8.1(a).	Backstep geometry	150
8.1(b).	Base grid for backstep	150
8.2.	$u(x,y)$ for $Re = 100$	151
8.3.	$v(x,y)$ for $Re = 100$	152
8.4(a).	$u(x,y = 0.0625)$ for $Re = 100$	153
8.4(b).	$u(x,y = 0.5625)$ for $Re = 100$	153
8.4(c).	$u(x,y = 0.8125)$ for $Re = 100$	154
8.4(d).	$u(x,y = 1.0625)$ for $Re = 100$	154
8.4(e).	$u(x,y = 1.5625)$ for $Re = 100$	155
8.4(f).	$u(x,y = 1.8125)$ for $Re = 100$	155
8.5(a).	$u(x = 1.0,y)$ for $Re = 100$	156
8.5(b).	$u(x = 4.0,y)$ for $Re = 100$	156

8.5(c).	$u(x = 6.0, y)$ for $Re = 100$	157
8.5(d).	$u(x = 12.0, y)$ for $Re = 100$	157
8.6(a).	$v(x, y = 0.25)$ for $Re = 100$	158
8.6(b).	$v(x, y = 0.75)$ for $Re = 100$	158
8.6(c).	$v(x, y = 1.0)$ for $Re = 100$	159
8.6(d).	$v(x, y = 1.25)$ for $Re = 100$	159
8.6(e).	$v(x, y = 1.75)$ for $Re = 100$	160
8.7(a).	$v(x = 1.0, y)$ for $Re = 100$	161
8.7(b).	$v(x = 4.0, y)$ for $Re = 100$	161
8.7(c).	$v(x = 6.0, y)$ for $Re = 100$	162
8.7(d).	$v(x = 12.0, y)$ for $Re = 100$	162
8.8.	Comparison of difference schemes; $u(x, \Delta x, y = 0.8125)$	163
8.9.	Comparison of difference schemes; $v(x, \Delta x, y = 1.25)$	164
8.10.	Comparison of difference schemes; error vs. Δx	165
8.11.	Solution error in u for $Re = 100$	166
8.12.	Solution error in v for $Re = 100$	168
8.13.	Truncation error in u for $Re = 100$	170
8.14.	Truncation error in v for $Re = 100$	172
8.15.	Rotated refined rectangles for backstep	174
8.16.	Notation for refined grids	174
8.17.	Convergence of active method on two-level grid system	175
8.18(a).	Effect of internal boundary location; $x_{L_1} = 4$	176
8.18(b).	Effect of internal boundary location; $x_{L_1} = 6$	176
8.19.	Reattachment length vs. Reynolds number	177

List of Tables

Table	Page
4.1. Refined Grids for Example 1	64
8.1. Parameters for Uniform-Grid Calculations	145
8.2. Uniform Grid Error	146
8.3. Inner Iteration Strategy	147
8.4. Adaptive vs. Uniform Grid Performance for $Re = 100$	148
8.5. Summary of Adaptive Backstep Calculations	149

Nomenclature

a_{ij}	Elements of matrix A.
A^u, A^v	Proportionality variables in Eq. (6.4.6).
c	Constant in Eqs. (6.1.8)-(6.1.10).
$c_{\epsilon_1}, c_{\epsilon_2}$	Constants in Eq. (6.1.12).
e_{rms}	rms solution error.
$e(h,x)$	Solution error.
$\tilde{e}(h,x)$	Solution error estimate.
G_k	Represents all grids at k-th refinement level.
$G_{k,j}$	j-th component of G_k .
h	Mesh size; step height.
H	Mesh size.
k	Time step; turbulent kinetic energy.
K	Ratio of mesh size to time step; sensitivity parameter in Eq. (A.8-9).
L	Differential operator; length scale.
L_h	Difference operator.
$L_{x,H}^p$	p-th order difference operator for x-momentum equation with mesh size H.
$L_{y,H}^p$	p-th order difference operator for y-momentum equation with mesh size H.
L^{-1}	Green's function.
L_h^{-1}	Inverse of L_h .
\dot{m}	Mass flux.
M	Matrix containing second moments of a set of flagged points.
M^x	x-derivative of pressure.
M^y	y-derivative of pressure.
\underline{n}	Outward unit-normal vector.

N	Number of grid points.
N_x, N_y	Number of mesh lengths in x and y directions, respectively.
p	Pressure.
p'	Pressure correction.
Pe	Peclet number.
Q	Volumetric flow rate.
Q_h	Difference operator.
R	Refinement ratio.
Re	Reynolds number.
$Re_{\Delta x}$	Cell Reynolds number.
Re_{eff}	Effective Reynolds number.
S_ϕ	Source term for scalar equation.
S_p	Source term for scalar pressure equation.
t	Time.
\underline{u}	Velocity vector.
u	x -component of velocity.
u_m	Maximum inlet velocity.
v	y -component of velocity.
u^*, v^*	Intermediate velocities calculated from Eq. (6.4.3).
u_h, v_h	x - and y -components of velocity on a grid with mesh size h .
V	Velocity scale.
V_o	Velocity outside of boundary layer.
x	Spatial coordinate.
x_{L_k}	x -location of downstream boundary of G_k .
x_R	Reattachment length.
\bar{x}_m, \bar{y}_m	Mean values of a set of x, y pairs.
x_1, x_2	Left and right grid boundaries.

y	Spatial coordinate.
y_0	Location of discontinuity at upstream boundary in Eq. (5.4.1).
y_1, y_2	Left and right boundary values.
w^1, w^2	Weighting functions in Eqs. (6.6.4).

Subscripts

e	East face of control volume.
h	Fine grid quantity.
H	Coarse grid quantity.
i	Row index.
j	Column index.
k	Refinement level.
L, R	Left and right boundary locations.
n	North face of control volume.
s	South face of control volume.
w	West face of control volume.

Superscripts

m, n	Iteration index
p, q	Order of accuracy.
x	x-momentum equation.
y	y-momentum equation.
$'$	Fluctuating quantity.

Greek Symbols

α_x, α_y	Masking arrays, Eq. (7.2.1).
δ	Boundary-layer thickness.
δ^*	Displacement thickness.
δ_{\max}	Maximum allowed estimated solution error.
ϵ	Diffusion coefficient (perturbation parameter); turbulent dissipation.

Γ_k	Boundary of G_k .
γ_k	Internal fine-grid boundary of G_k .
Ω_k	Domain of G_k .
$\tau(h,x)$	Truncation error.
$\tilde{\tau}(h,x)$	Truncation error estimate.
τ_{\max}	Maximum allowed truncation error.
ω	Relaxation factor in Eqs. (6.4.10)-(6.4.11).
ϕ	Passive scalar.
ϕ_e	Exact solution to Eq. (5.2.1).
$\sigma_k, \sigma_\varepsilon$	Constants in Eqs. (6.1.11)-(6.1.12).
θ	Angle of rotation.
ξ, η	Rotated coordinates.
ρ	Fluid density.
ν	Kinematic viscosity.
ν_T	Eddy (turbulent) viscosity.
ν_{eff}	Effective viscosity.
δ/δ_x	Generalized difference operator for first derivative.
$\delta^+/\delta\xi$	Backward difference operator for first derivative.
$\delta^-/\delta\xi$	Forward difference operator for first derivative.
$\delta_{\xi\xi}, \delta^2/\delta x^2$	Difference operators for second derivative.
$\Delta\phi_{\text{sz}}$	Stopping criterion for Schwarz iterations.
$\Delta\phi_{\text{GS}}$	Stopping criterion for Gauss-Seidel iterations.
∇^2	Laplacian operator.
∇	Gradient operator.
Δx	Mesh size in x-direction.
Δy	Mesh size in y-direction.

Other Symbols

$(-)$	Average value.
-------	----------------

Chapter 1

INTRODUCTION

1.1 Motivation

Computational fluid dynamics (CFD) is playing an increasingly important role in the design and analysis of energy-conversion and transportation systems, due to the development of solution algorithms that are efficient and accurate and to the rapid advances made in computer technology. However, there is always a need for more efficient algorithms. Design studies require repeated calculations of a given problem in order to search a parameter space, and engineers are continually tackling more complex and hence computationally more challenging problems.

Many of the flows encountered in these systems are governed by elliptic partial differential equations whose solutions may exhibit fine structure within small regions of the computational domain. Many traditional solution techniques require a fine mesh covering the complete domain in order to resolve these fine local details. This method is inefficient, since the fine mesh is not needed in parts of the flow where the solution has a moderate variation. In some cases, this inefficiency can be tolerated; in others, it can be prohibitively expensive.

A lack of resolution is also a hindrance to the development of turbulent closure models. Kline et al. (1982) point out that it is difficult to distinguish the numerical errors from modeling errors, and they call out for procedures that can guarantee a prescribed level of numerical accuracy.

Adaptive grids can simplify solutions to problems that need refinement only in small, localized regions of the domain. In these methods, the mesh is changed or adapted as the solution develops. The mesh is adjusted or refined to accurately resolve fine structures as they appear in the numerical solution. The result is a nonuniform distribution of grid points that provides the desired level of accuracy at much lower cost than a uniform fine grid.

In the remainder of Chapter 1, we review existing adaptive grid methods. We then review the technique developed by Berger (1982) for hyperbolic equations. We have applied this approach to elliptic problems. Chapter 2 describes Berger's method in some detail. In Chapter 3 we present the strategy for applying Berger's method and the issues particular to elliptic equations. Chapter 4 describes the application of our method to linear, two-point, boundary-value problems. In Chapter 5, results and a performance evaluation are given for a two-dimensional, linear, convection-diffusion problem.

We discuss the basic solution technique and pertinent issues for the Navier-Stokes equations in Chapter 6. Chapter 7 describes the features of our adaptive Navier-Stokes solver. In Chapter 8, we present numerical results, including a performance evaluation for the method applied to the laminar, backward-facing-step problem. We draw conclusions and make recommendations for further improvements of the method in Chapter 9.

1.2 Adaptive Grid Methods

The development of adaptive grids is probably the most important area of research in grid generation at present (Thompson, 1984). Many approaches have been developed for a variety of differential equations and applications. These include steady and unsteady problems, hyperbolic, parabolic, elliptic, mixed equations, etc. It would be difficult to make a classification for all adaptive grid methods, but most methods can be fit into two different categories.

The first category is the "moving mesh" technique. Here, the total number of grid points is fixed, and the mesh is adjusted by moving the points away from regions that have small solution variation and towards regions having large variation. The method of moving the points and the criteria for determining where they go generally differentiate methods in this category. These methods can also be classified as "global" refinement techniques, since the complete computational domain is usually involved in the adaptation.

In the second strategy, mesh points are added (or deleted) as needed in order to give the desired solution accuracy; the total number

may change. However, the addition or deletion is local in nature, and we therefore refer to these techniques as "local" mesh-refinement methods. Again, the number of ways in which points are added or deleted makes many variations of this approach possible.

We next discuss some of the existing global refinement strategies and discuss their advantages and disadvantages. We follow this with a similar presentation for some local refinement strategies. Further discussion of adaptive grid methods can be found in recent review articles by Thompson (1983), Anderson (1983), and Hedstrom and Rodrigue (1982). Note also that we do not discuss adaptive techniques for finite element methods. Babuska et al. (1983) have given a recent review of finite-element adaptive-grid methods.

1.2.1 Global Refinement Methods

Most global refinement methods are used in conjunction with grid-transformation methods. In these methods, the grid is nonuniform in physical space and is generated by mapping the irregular physical domain into a rectangular computational domain on which a uniform grid is used. The solution is calculated in the computational space and transformed back to the physical coordinates. The grid transformation results in modifying the differential equations; in particular, derivatives are multiplied by gradients of the mapping function called metrics. In physical space, the grid is usually boundary-conforming.

For problems in complex geometries, numerical techniques are simplified if a transformed grid is used. The uniformity of the computational grid makes the programing and data storage very straightforward. Furthermore, a solver written for the uniform rectangle can be applied to a variety of problem geometries and grid-point distributions.

Using a fixed number of grid points and an assumed initial distribution of points in physical space, global refinement methods adjust the grid transformation as the solution develops. Global methods differ from each other in the way in which the transformation is generated and updated, and in the criteria used to drive the adjustment.

There are two subclasses of adaptive transformation methods. When time accuracy is required, the transformation is time dependent, and the mesh is adjusted at every time step. In the second class, the transformation is time independent, and the mesh is held fixed for a given number of iterations before it is modified.

For time-dependent problems, the grid speed appears in the transformed differential equations; the adaptation criterion is based on an auxiliary equation for the grid speed. The new positions of the grid points are calculated from the grid speeds. Given the new point locations, the metrics are then reevaluated.

For time-independent problems, the grid speed does not appear in the differential equations. Adaptive methods of this type calculate the new grid-point locations directly. The solution is interpolated onto the new mesh and new metrics evaluated before another solution is generated.

Certain restrictions must be observed in either of these techniques. Grid points should concentrate in regions of rapid solution variation, but no region should become void of points. The point distribution should be smooth, and in two or three dimensions grid lines should not become too far from orthogonal (Mastin, 1982).

Brackbill (1982) discusses a method that uses a variational formulation which explicitly addresses these restrictions. The method minimizes a linear combination of three functionals of the grid. The first functional is a measure of the rate of change of the grid spacing. It is intended to control the smoothness of the distribution. The second is a measure of the nonorthogonality of the mesh lines. The third functional is the integral of the product of a specified weighting function and the mesh cell volume. The weighting function is intended to measure the solution error or variation and make the mesh spacing inversely proportional to the error or variation.

Saltzman and Brackbill (1982) applied this variational method to a 2-D, time-dependent solution of the Euler equations having multiple shocks. The number of times the mesh was adapted during the simulation was not given. The weighting function used to control mesh spacing was

the pressure gradient. The authors obtained what appears to be a nearly optimal grid at steady state, but they do not discuss the solution accuracy, the adaptive efficiency, or the computational work.

This method has two major drawbacks: computational expense and use of heuristic adaptation criteria. The solution of the variational problem at each adaptation costs a large fraction of the cost of generating the solution itself. Also, the weighting function used to drive the mesh adaptation is not a direct measure of solution error. Further, the linear combination of the three functionals was determined by trial and error and may not be appropriate to all cases.

Pierson and Kutler (1980) also used a variational formulation as a basis for an adaptive grid method. The transformation from physical to computational space is specified as a linear combination of Chebyshev polynomials. For the methods they used, the leading term in the truncation error is proportional to the third derivative of the solution. Thus, the integral of the square of a finite difference approximation to the third derivative is minimized; it is constrained by limiting the maximum and minimum mesh sizes. The minimization problem yields the coefficients for the polynomials of the transformation.

The method is applied to steady, one-dimensional, boundary-value problems. The procedure is begun by calculating an initial solution on a uniform grid. A new grid is then generated by solving the minimization problem. Finite difference estimates of the third derivative are calculated using the initial solution. A new solution is then calculated on the adapted mesh. The accuracy of these two solutions was assessed by comparing them to a calculation done on a uniform mesh having twice as many grid points; the adapted solution had better accuracy. No attempt was made to refine the mesh further. The authors also applied the method to a time-dependent problem. The mesh had to be frequently updated to maintain good accuracy.

The advantage of the Pierson-Kutler method is that it tries to minimize a measure of the solution error. However, the computational expense of solving the minimization problem is considerable, especially for time-dependent problems.

A grid speed based method is discussed in Rai and Anderson (1981a,b) and Anderson and Rai (1982). The auxiliary equation used to drive the grid point motion is formulated to equi-distribute an arbitrary quantity over the mesh. They use a point electrical charge analogy; local variations cause points to attract or repel each other. The intent is to use a truncation-error estimate as the quantity to be equi-distributed; however, the gradient of a dependent variable has been used in all their examples.

The method has been applied to steady and unsteady problems in one and two dimensions, primarily to problems containing shocks. The method clusters points where the solution gradients are strong; however, errors were introduced by stretching the grid too much in low-gradient regions. Also, grid speeds had to be limited; otherwise oscillations in the grid occurred. To overcome these difficulties, empirically determined parameters and a grid-speed damping relation were introduced.

An obvious difficulty with this method is its use of problem-dependent empirical factors. A second difficulty is the use of the solution gradient as an error indicator. This is sufficient if all the error is incurred at shocks, but it is inadequate if the solution has significant higher-order derivatives elsewhere. Noting that finite difference estimates of truncation error are generally very noisy, Rai and Anderson point out that smoothing of these estimates is required.

Dwyer et al. (1980, 1982) also discuss a method in which the maximum change in the solution between grid points is the criterion for refinement. The criterion also includes the change in the gradient between points. Use of equal weighting of these two quantities on a uniform grid is equivalent to equi-distributing a weighted average of the solution gradient and curvature.

The formulation of the grid transformation is time-dependent, but the grid speed is not explicitly calculated. Rather, the mesh is adjusted, the new metrics evaluated, and the grid speed determined from the change in the position of the grid points.

The method is applied to combustion problems, some of which have moving flame fronts. Both elliptic and parabolic problems were solved.

In two dimensions, only one of the coordinates is adjusted. Thus, the method is essentially a one-dimensional adaptive technique.

The results of a uniform grid solution of a two-dimensional flame-propagation problem in cylindrical geometry are compared with a similar calculation done using a grid adapted in the radial direction. The uniform grid solution deteriorated as the flame moved towards larger radii, where mesh size is larger. Oscillations and negative temperatures developed, due to the loss of resolution, and the calculation had to be terminated. The adaptive grid maintained accuracy at the larger radii.

Dwyer et al. implied that their scheme can be extended to two dimensions, but this has not been done. Problems will arise because there is no control of grid skewness. Also, use of problem-dependent parameters to control the adaptation is a disadvantage. The authors state that attempts to base the adaptation on estimates of higher-order derivatives led to grid instabilities. This seems to be characteristic of most global refinement methods.

Gnoffo (1982, 1983) discusses a time-independent transformation method and applies it to the Navier-Stokes equations. Like the previous method, the grid adaptation is one-dimensional and the solution gradient is equi-distributed. A spring analogy is used to formulate the adaptation criteria. Grid points along one coordinate direction are assumed to be connected by springs whose constants are proportional to the local gradients.

This method suffers from problems similar to those discussed above. Complex flowfields cannot be handled, since the adaptation is in only one dimension. Smoothing and damping of the grid adjustment has to be included for stability. Additionally, grid skewness is not controlled.

Nakahashi and Diewert (1984, 1985) improve upon this spring analogy method. They extended the method to two- and three-dimensional problems. Grid points are connected to adjacent points with tension springs whose constants are proportional to the local solution gradient. Additionally, torsion springs are connected to each grid point to control the inclination of the coordinate lines, thus preventing excessive skewness.

The method uses an efficient technique for updating the grid. The procedure is split into a sequence of one-dimensional adaptations. Three-dimensional grid adjustment is achieved by the successive application of the one-dimensional scheme. The coupling of information is constrained to be one-sided, allowing a marching solution procedure to be used in the one-dimensional adaptations.

The method is applied to steady, supersonic flow problems in two and three dimensions. The resulting adapted grids appear to be of good quality for the problems solved, which possess complex flow and shock fields. It is not surprising that the method worked well for these flowfields, since grid and solution can both be marched in the same direction. It is not clear whether the efficiency will be maintained for problems that are elliptic; however, the technique is probably the best of the current time-independent refinement methods.

Greenburg (1985) describes a grid-speed method based on a chemical reaction analogy. The time rate of change of a local mesh length is made proportional to its neighboring mesh lengths multiplied by reaction-rate constants. These constants are based on formulas specified to produce the desired adaptation criteria, including equidistribution of solution gradient, grid smoothness, minimum mesh length, etc. The method is implemented and applied in one dimension only for a time-dependent linear problem. Application to higher dimensions is forthcoming.

To conclude our discussion of global refinement techniques, we mention the work of Pearson (1981). Although his method is not adaptive, his goal is to compute the streamlines in steady, inviscid, compressible flow. It is mentioned here because the purpose of many adaptive grid methods is to produce an optimal or nearly optimal grid. For fluid-flow problems, the stream- and velocity-potential lines are a nearly optimal coordinate system, since the differential equations are greatly simplified and the numerical error associated with the flow being oblique to the grid lines is zero. Additionally, one wonders whether these coordinates should be the goal of any adaptive grid method in CFD.

Pearson recasts the momentum and continuity equations in terms of the x-coordinate and two streamline parameters lying in a plane normal to the x-axis. The parameters are constant along a streamline. This method is restricted to flows that have no streamlines perpendicular to the x-direction. For supersonic flows, the solution and streamlines are marched in the x-direction. For subsonic flow, an iterative solution procedure is used, similar to shooting methods for boundary-value problems. The author points out that the method can have stability problems, particularly if there is large curvature in the streamlines.

The determination of the streamlines as coordinates, along with the velocity and pressure fields, is really practical only for a small class of flows, most notably inviscid, irrotational flows. The effort needed to calculate the coordinate system for complex flows is hard to justify.

To summarize, global adaptive-refinement techniques are generally suited for use with grid-transformation solution methods, since they are formulated to adjust the transformation. The distribution of a fixed number of grid points is adjusted during the solution/grid iteration process.

A major shortcoming of this type of refinement is the use of heuristic adaptation criteria. The basis of a natural refinement criterion is the local truncation error. It causes the error in numerical solutions. By minimizing the truncation error, the solution error can be minimized. However, the truncation error is noisy because it is a combination of higher-order derivatives. Numerical estimates of it are noisier. Consequently, global refinement methods must resort to using smoother adaptation criteria, such as equi-distribution of solution gradients. However, the gradient can be a misleading error indicator. For example, a steep linear function can be exactly represented with only two grid points!

Also, these methods must employ arbitrary parameters and smoothing functions to maintain grid quality. Quality is assessed in terms of smoothness of the point distribution, skewness of grid lines, proper clustering in high truncation-error regions, minimum point distribution in low-error regions, etc. The disadvantage of using arbitrary

functions is that they are often problem-dependent and have to be determined by trial and error.

The expense of performing the global adaptation is proportional to the total number of points in the grid. We shall see that this expense can be reduced by locally refining the grid.

1.2.2 Local Refinement Methods

The second class of adaptive grid techniques contains the local refinement methods. Grid points are added to (or deleted from) the global grid where some measure of the solution error is large (or small). Since the regions of large error are usually localized in space, the resulting refinement is applied locally. The solution is recalculated on the new grid, and the refinement process can then be repeated. Iterative improvement of the grid and solution effectively equi-distributes the measure of the error.

There are two advantages of local refinement over the global methods discussed earlier. The locations of the existing grid points do not have to be updated; only those of the added points need to be accounted for. This lowers the overhead for performing the grid adaptation. Secondly, since the existing grid points are static, noisy error measures can be used to cluster grid points without giving rise to the instabilities of the moving-grid methods. Thus, the natural refinement criterion, the truncation error, can be used as the error measure in these methods.

Local refinement methods can be further broken into two categories, depending on the way in which grid points are added. In the first category, points are inserted or "embedded" into the existing grid structure and a single grid covers the problem domain at any one time. In the second category, the refinements are "overlaid" on top of the base grid. We next discuss some embedded mesh methods found in the literature, followed by a discussion of some of the overlaid methods.

Dwyer et al. (1982) describe a local refinement method similar to their global method discussed in Section 1.2.1. The same refinement criterion, equi-distribution of a linear combination of solution gradient and curvature, was used for the local method.

In this method, an initial grid is specified and a solution is calculated on it. Then, for one-dimensional problems, a grid point is added between two existing points if the refinement criterion is violated. For two-dimensional problems on rectangular grids, they add a grid line. (They can remove grid points or lines if the local criterion falls below minimum specified values.) The solution is interpolated onto the new grid points to provide an initial guess for solving on the new grid. The procedure is iteratively applied until the maximum gradient and curvature in the field fall below the maximum specified values.

The method is applied to a one-dimensional, steady-flame problem described by 72 ordinary differential equations. The grid points are added in the flame-front region, where there are large changes in the solution components. The adaptive calculation is seven times faster than a uniform-grid calculation having similar accuracy.

The technique is also applied in two dimensions to a nonlinear, elliptic problem. Since grid lines are inserted between points where the criterion is violated, additional grid points are added where they are not needed, decreasing the adaptive grid efficiency. In problems in which the region needing refinement is long, narrow, and oblique to the grid lines, the whole grid will be refined.

Murman and Baron (1983) discuss an adaptive, embedded mesh scheme for the Euler equations to be used in conjunction with a multigrid method. They do not recommend a specific refinement criterion, but suggest several possibilities, including solution gradient and second derivative. Mesh cells are subdivided if the refinement criterion is exceeded. A pointer system similar to the type used in finite element methods keeps track of storage locations for solution values. The authors point out the need for maintaining conservation and accuracy everywhere.

Murman and Baron's method is applied to one- and two-dimensional problems. Savings of a factor of 2-3 in computer time compared to a uniform, fine-grid calculation are obtained. A disadvantage of the method is that multiply connected, embedded meshes containing "holes" can result. Regions needing refinement should be contiguous and not

have holes. The presence of a hole could arise from the use of an inadequate error indicator. Not refining a region that should be refined may stall the error-reduction ability of an adaptive refinement process. It would be safer to plug all holes. An additional disadvantage of the method is the difficulty of vectorizing the algorithm due to the irregular data-storage technique.

Bai (1984) investigates embedded refinement using multigrid methods, as originally suggested by Brandt (1977). Working with Poisson's equation in two dimensions, Bai specifies local refinements a priori, so that his method is not really adaptive. However, he states that implementation of automatic refinement is straightforward. As in the previous method, the grid is refined by subdividing mesh cells.

A procedure is derived for optimizing the grid refinement. The goal is to minimize the solution error for a given amount of computational work. The criterion for refinement thus becomes a complex function of an estimate of the local truncation error, a spatial error-weighting function, and a function describing the computational work. Bai also discusses the issues of interpolation and conservation at the embedded grid interfaces.

Optimal refinement can be considerably more expensive for problems more complicated than Poisson's equation. In these cases, it may be more practical to simply search for a refinement that gives adequate accuracy. Alternatively, a crude implementation of Bai's optimization scheme could be implemented. Since multigrid methods are very efficient, Bai's work is important; however, the development of practical refinement and data-management procedures is required.

Brown (1982) discusses an adaptive grid method that is an extension of a technique described in Kreiss and Kreiss (1981). Two-point boundary-value problems for a system of singularly perturbed, linear, ordinary differential equations are considered. The results are applied to semi-discretized partial differential equations.

The solutions to singular perturbation problems admit internal and boundary layers. Brown develops a theory that facilitates adaptive solution of these systems under certain constraints. It is shown that the solution error can be reliably estimated in terms of lower-order

divided differences, primarily approximations of the first and second derivatives. These differences are used as error estimates to drive an adaptive grid procedure.

The numerical method used is crucial to the success of this method. It is well known that using central differences for these problems may produce solutions that oscillate wildly over the whole computational domain if the mesh size is too large to resolve the thin boundary and internal layers. Use of first-order difference methods (upwind differences) gives good solutions outside these layers, but grossly enlarges the thickness of the thin layers. To obtain useful information from initial coarse-grid solutions, it is necessary to use the first-order methods. This prevents the unnecessary overrefinement that an oscillating solution would require. Higher-order differences can be employed once the mesh size in the vicinity of the thin layers is small enough. Brown therefore uses a hybrid scheme that smoothly switches between central and upwind differences, depending on the local mesh size and the resolution of the thin layers.

The method is demonstrated on one-dimensional, stationary, and moving shock problems. Results are also given for two-dimensional problems in which the shocks are oblique to the coordinate system. A splitting technique is used for these calculations, in which the one-dimensional adaptive procedure is applied along the coordinate lines in each direction. A nonuniform distribution of grid points in the regions of the shocks is generated by adaptive refinement. The resulting shock profiles are very sharp and accurate.

This method appears to be very promising, especially for shock calculations. Further development may be required to extend the method to viscous flows. The notion of using difference methods that can generate smooth solutions to provide useful information for further refinement is an important one for adaptive grid methods in general. A drawback of the method is the nonuniform data structure that accompanies embedded refinements.

We finally come to the overlaid type of local grid refinement. This technique has been introduced and developed primarily by Olinger and his co-workers Berger, Bolstad, and Gropp for the solution of hyperbolic

equations. We next summarize the principles behind the method as elaborated in Olinger (1984). We then follow with a summary of the applications of the method.

The primary goal of this solution-adaptive technique is to achieve the desired solution accuracy at near-minimal cost. The cost includes both computer and program-development expenses. To satisfy this goal, piecewise regular grid structures were selected. More grid points are used than in the global methods, but there is less overhead (computational work and storage) per point, due to the regularity of the refinement. The search for an optimal grid is abandoned in favor of a good enough grid.

A second aim is to relate the grid to the desired accuracy. This can be contrasted with the global refinement methods in which the adequacy of the results is not addressed very well. This goal is realized by basing the refinement on asymptotically accurate estimates of the solution and truncation errors.

The procedure begins with the generation of a solution on an initial coarse grid. The truncation error is then estimated using a variant of Richardson extrapolation, which we describe later in more detail. Points having large estimated error are "flagged". These points are then separated into spatially distinct clusters which define local refinement regions. These regions are then fit with local, overlapping rectangles of arbitrary rotation in a manner that minimizes the size of the refined region. Each rectangle is given a uniform grid. The initial and boundary values for these grids are interpolated from the coarse grid. Each refined grid is treated independently and possesses its own regular data storage. The solution is recalculated on this new grid system, and the process can be repeated. New levels of refinement are overlaid on the existing grid system. This iterative improvement of the grid and solution is repeated until the maximum truncation error in the domain falls below a maximum specified value. The overall result is an equi-distribution of the error.

Gropp (1980) first demonstrated the feasibility of the technique for a two-dimensional hyperbolic problem. He used one level of refinement. The solution is advanced from time t to $t + dt$ on the coarse

grid. Mesh cells are then subdivided where the local gradient exceeds a specified value. Values at time t are interpolated onto the fine mesh from the coarse mesh. These are then advanced to time $t + dt$ on the fine mesh. Coarse grid points underlying fine grid points are then assigned the corresponding fine-grid solution values. The fine grid is then discarded, and the procedure repeated again. In this way, the fine grid follows moving regions of large gradients. Gropp's adaptive calculation is consistently twice as fast as a uniform-grid calculation having the same accuracy, even though one third to one half of the region is refined.

Bolstad (1982) extended the adaptive procedure to an arbitrary number of levels of refinement. He applied the technique to systems of hyperbolic equations in one space dimension. Finer refinements are spatially and temporally nested within the previous refinement. Refined grids can be created, destroyed, merged, and separated, permitting the refinement to follow moving discontinuities without actually moving each grid. Local truncation error was estimated using Richardson extrapolation in order to determine where refinement was required. The procedure for solving and updating the grid refinement was similar to that used by Gropp, except that it was generalized to an arbitrary number of refined levels.

Bolstad made an adaptive calculation for the wave equation. The exact solution was two counter-streaming Gaussian pulses superimposed on a sinusoid. The refinement followed each pulse as it entered the domain, merged, and separated from the other pulse. The method also performed well for a shock-tube calculation. The method was evaluated by comparing with results on uniform grids having the same mesh size as the finest level of refinement in the adaptive calculation. The adaptive calculation was 3-5 times faster. Additionally, there was a 50% savings in storage.

Berger (1982) extended the adaptive method to hyperbolic equations in two dimensions. The grid refinements are rectangles of arbitrary orientation in space. At a given level of refinement, the rectangles are allowed to overlap. The use of such rectangles allows the local coordinate system to be approximately aligned with flow features, reduces the size of the refined region, and requires very little overhead to

maintain. The adaptive solution procedure was similar to Bolstad's. Berger utilized nontraditional data structures to keep track of the various grids. We discuss them in Chapter 2; a detailed description can be found in Berger (1983). The local truncation error was estimated using Richardson extrapolation, and provided the criteria for adaptation.

The method was applied to linear and nonlinear problems in one- and two-space dimensions. Adaptive computations ran 4-7 times faster than uniform-grid calculations of similar accuracy. In Berger and Jameson (1985), the method is applied to the Euler equations. Steady-state calculations of transonic flow about airfoils ran faster than uniform-grid computations by factors up to 20.

We can now summarize the advantages of the overlaid, adaptive-grid refinement technique. The use of piecewise, uniform refinement provides for efficient utilization of storage and processor time. Significant speed-up in calculational times has been demonstrated. The method permits the use of parallel computation. Use of rotated rectangles allows the coordinate system to align with the flow and flow features. This minimizes the numerical diffusion error that occurs when the flow is oblique to the grid lines. Since all grids are uniform rectangles, the user needs only to provide a single standard solver to the adaptive program. Finally, the accuracy of a calculation is explicitly assessed and controlled.

This adaptive approach is not without shortcomings. One disadvantage is that the computer programming is much more complicated. However, once the adaptive part of the program has been written, the flow solver and boundary conditions can usually be changed quite easily. Another disadvantage is that interpolation is required to communicate solution information between grids. High-order accurate interpolation is desirable, but its implementation is cumbersome. Additionally, special care must be taken with the treatment of internal boundaries, where it is important to maintain accuracy and conservation.

Because of its favorable characteristics, we decided to apply the overlaid, adaptive-refinement approach to elliptic flow problems. We review Berger's method in detail in the next chapter. The chapter that follows discusses how we applied the technique to our elliptic problems.

Chapter 2

REVIEW OF BERGER'S METHOD

2.1 Overview

In this chapter, we review Berger's adaptive method in order to explain this approach in some detail. The chapter also serves to highlight the specific ideas used in the development of our adaptive method for elliptic equations. A more complete description of the method can be found in Berger (1982) and Berger & Oliger (1984).

Berger's method was developed for finite difference solution of systems of hyperbolic equations in one and two space dimensions, with explicit time differencing. It was designed for problems in which the solutions are locally irregular, but the boundaries are simple. It does not deal with complex geometry.

The grid is refined locally in space and time. The approach is to generate independent, refined subgrids as needed to cover the irregular region(s) of the solution. In two dimensions, the subgrids are rectangles of arbitrary orientation. The solution on each subgrid is approximated by the same finite difference method as on the original (base) grid. The regions in which the solution is irregular change in time; the subgrids are allowed to follow them. The algorithm makes no assumptions about the size or shape of refinement regions, nor their directions or speeds.

A description of the grid system is given in the next section, followed by discussions of the adaptive solution procedure, the error estimation and refined grid-generation techniques, and the treatment of boundary and initial values. We finish with a discussion of the data structures.

2.2 Grid Description

The initial, coarsest grid is specified by the user. This base grid is denoted G_0 and remains fixed during the computation. The base grid may be a single grid covering the computational domain, or it may be a union of several, possibly overlapping, component grids. If there

are several components, a typical one is denoted $G_{o,j}$. The base grid G_o is then the union of the component grids. Each component grid is required to be uniform in some coordinate system. (A uniform, rectangular grid has constant mesh spacing in the two coordinate directions.)

Component grids are treated independently, each having its own solution vector, storage area, coordinate system, etc. Because of this independence, the algorithm is a domain-decomposition method. It permits separate processing of each grid, including solution generation, updating of boundary conditions, etc. Domain decomposition also allows separate parts of the domain to be approximated by different differential and difference equations although this was not done in Berger's program. This was done in Bolstad's work, however. This approach is called zonal modeling in the engineering literature.

Figure 2.1 shows an example of a base grid made up of two component grids, a curvilinear, boundary-conforming grid and a uniform rectangular one. If the curvilinear grid is mapped to a computational space, both grids can be uniform rectangles. Calculations on overlapping component grids have been previously done by Starius (1977), Atta and Vadyak (1983), and Dihn, Glowinski, and Periaux (1984).

Subgrids having smaller mesh sizes are generated during an adaptive computation. They are overlaid on top of the coarser grid(s), covering the regions needing refinement. In two dimensions, the subgrids are uniform rectangles having arbitrary orientations. Using uniform grids minimizes the storage required for grid point locations, allows the use of more accurate difference formulas, and provides for efficient solution procedures. The advantages of the rotation have been pointed out previously. Subgrids are also treated independently.

Subgrids are allowed to contain even finer subgrids. Thus, a hierarchy of "levels" of grids is constructed during the adaptation process. The coarse grid, G_o is at the level 0 in this hierarchy. Subgrids of G_o , denoted G_1 , are at the level 1 refinement. Subgrids of G_1 at the level 2 refinement are denoted G_2 , and so on. Subgrids of G_k are constrained to be wholly contained within G_k 's boundaries. The resulting grid structure therefore becomes a nested sequence of finer

and finer meshes. An example grid structure is illustrated in Fig. 2.2, where a component grid of level k refinement is denoted $G_{k,j}$.

The mesh size for all grids on level k is specified as a constant multiple of the mesh size on level $k + 1$, called the refinement ratio. Typical values are 2 and 4, although a value of 10 has been used in some cases.

2.3 Adaptive Solution Procedure

There are three main tasks in the adaptive solution process: (1) time advancement of the solution on the current grid, (2) error estimation and refined subgrid generation, and (3) inter-grid communication. In this section we discuss the time-advancement method and how the adaptive procedure is executed. Error estimation and grid generation are covered separately in sections that follow.

We describe the time-advancement procedure by first assuming that we have an existing grid structure (e.g., the grid shown in Fig. 2.2). Initial and boundary values are also assumed specified for each grid. A time-explicit difference method is used to advance the solution one time step. Because each grid is treated independently, it is merely necessary to specify the order in which the individual grids are advanced. For implicit methods, a different technique must be employed.

The order of integration is related to the time step used for each grid. The ratio of time steps for consecutive grid levels is set equal to the refinement ratio, R , of the mesh sizes. This makes the ratio of mesh size to time step, K , constant for all grids, and is appropriate for hyperbolic equations. Specification of different time steps for each level provides additional efficiency, since time steps on coarse grids are not limited by those on fine grids.

The order of integration becomes straightforward using a constant K . For every time step on level 0, the grids on level 1 are advanced R time steps, grids on level 2 are advanced R^2 time steps, and so on. The basic time unit is one coarse grid time step. All grids must be advanced to the base grid's time before another base grid step is taken. Figure 2.3 illustrates this procedure in one space dimension and

time, with $R = 2$. The order of advancement from coarsest to finest, for one coarse grid time step, is as follows (reading from left to right):

$$\begin{array}{ccccc} G_{0,1} & & & & \\ & G_{1,1} & & G_{1,1} & \\ & & G_{2,1} & G_{2,1} & G_{2,1} & G_{2,1} \end{array}$$

The error estimation and regridding procedures are the second major tasks that are performed in the adaptive process. Every several base-grid time steps, the error is estimated at all points in the grid. (The interval between error estimates is specified a priori.) A new fine grid can be created at this time. Initial values for new grids are obtained by interpolating from the finest grids in the current grid structure. Existing grids that are no longer needed can be removed by releasing their data-storage locations. These operations produce the adapted grid.

The last operation in the process provides the necessary communication between the grids, and consists of three sub-tasks. First, since subgrids usually have boundaries in the interior of the problem domain, boundary values have to be calculated for each. Values are obtained from either an overlapping fine grid at the same level or from a grid at the next coarsest level. Special care needs to be taken in the evaluation of these values. A more detailed discussion of the treatment of boundary values is given in a later section.

The second task is updating. Whenever a grid and its subgrid are integrated to the same time, the solution at common points in the coarse grid is replaced with that of the fine one. The purpose is to maintain accuracy on the coarse grids and is sufficient because the time advancement is explicit. Updating also provides the influence of the "inner" fine grid on the "outer" coarse grid solution.

The last intercommunication task is averaging and is performed only for overlapping fine grids at the same level. Averaging is required, because solution values on two grids at the same level may differ in the

overlapping region. The solutions on each grid in the overlap are replaced by the averaged values.

To summarize the adaptive procedure, the solution is advanced for a specified number of coarse-grid time steps. The error is then estimated, and the grid is adapted. The special grid intercommunication procedures are performed during both of these processes. After the grid is adapted and initialized, the solution can then be advanced again.

2.4 Error Estimation

Subgrids are placed over regions that need refinement. As stated earlier, a grid is refined where the truncation error is large. In this method, a variation of Richardson extrapolation is used to estimate the truncation error. In this section we show how the truncation error estimate is calculated and point out the advantages of this technique.

We begin the discussion by first introducing some notation. Consider a hyperbolic differential equation,

$$u_t = L[u] \quad (2.4.1)$$

where L is the spatial differential operator. A simple explicit finite difference method for this equation is:

$$\frac{u(x, t+k) - u(x, t)}{k} = L_h[u(x, t)] \quad (2.4.2)$$

Here, L_h is the spatial finite difference operator for a grid with mesh size h . This can be rewritten in a compact form as:

$$u(x, t+k) = Q_h[u(x, t)] \quad (2.4.3)$$

The truncation error for the difference method is obtained by substituting the exact solution to (2.4.1) into the finite difference equation (2.4.2) or (2.4.3). If the exact solution is smooth in space and time, the truncation error is:

$$\begin{aligned} u(x, t+k) - Q_h[u(x, t)] &= k(k^q a(x, t) + h^q b(x, t)) + kO(k^{q+1} + h^{q+1}) \\ &= \tau + kO(k^{q+1} + h^{q+1}) \end{aligned} \quad (2.4.4)$$

where τ is the leading order term. Note that the order of accuracy of the method in space and time are the same and equal to q . Taking two consecutive time steps with the method gives

$$u(x, t+2k) - Q_h^2[u(x, t)] = 2\tau + kO(k^{q+1} + h^{q+1}) \quad (2.4.5)$$

Let Q_{2h} represent the same difference method as Q_h except with mesh size $2h$ and time step $2k$. The truncation error for the $2h - 2k$ method is:

$$\begin{aligned} u(x, t+2k) - Q_{2h}[u(x, t)] &= 2k \left[(2k)^q a(x, t) + (2h)^q b(x, t) \right] \\ &\quad + kO(k^{q+1} + h^{q+1}) \\ &= 2^q(2\tau) + kO(k^{q+1} + h^{q+1}) \end{aligned} \quad (2.4.6)$$

Neglecting the higher-order terms in (2.4.5), subtracting (2.4.6) and dividing by $2(2^q - 1)$ gives:

$$\frac{Q_h^2[u(x, t)] - Q_{2h}[u(x, t)]}{2(2^q - 1)} = \tau + kO(k^{q+1} + h^{q+1}) \quad (2.4.7)$$

(2.4.7) provides an estimate of the leading term in the truncation error.

This is equivalent to advancing the solution two steps from time t with the standard method and comparing it with the solution obtained by taking one double-step on a $2h$ mesh. This is illustrated schematically in Fig. 2.4 for a simple explicit method which uses $u(x_h, t)$, $u(x, t)$, and $u(x+h, t)$ to evaluate $u(x, t+k)$.

A major advantage of this technique is that the exact form of the truncation error does not need to be known. For many differential equations, especially systems, the exact truncation error can be very complex and tedious to derive. The method is also independent of both differential and difference equations and therefore can be applied to a wide variety of problems without difficulty, in contrast to global refinement techniques which use heuristic error measures that are problem-dependent. The error estimation is also relatively inexpensive.

When it is time to estimate the error, (2.4.7) is evaluated at every point in the grid. If the pointwise truncation error estimate is greater than a prescribed value, the point is "flagged" to denote that refinement is needed in its vicinity. Once all the local error estimates have been calculated and checked, the collection of flagged points is then processed to generate the next level of refined subgrids.

2.5 Refined-Grid Generation

This section describes the procedure for generating refined grids to enclose a collection of flagged points. We discuss only the procedure used in two dimensions, as it is trivial in one dimension.

Grid generation is done in two steps. Flagged points are separated or clustered into spatially distinct groups. Individual clusters are then "fit" with the rectangles of arbitrary orientation. The "goodness of fit" is evaluated for each rectangle. If a rectangle has a bad fit (encloses too much of an area not requiring refinement), it is broken into subclusters that are refit with new rectangles.

Clusters are created with an algorithm which requires all points in a cluster to be near neighbors. A new cluster is begun by assigning one point to it. Other points are added to the cluster if their distance from any point in the cluster is less than a specified value. The intercluster distance is a small integral number of mesh widths.

Each cluster is then fit with an ellipse determined in the following manner. Let matrix A be the $n \times 2$ matrix of the coordinates of the points relative to their mean (x_m, y_m) , n the number of points in the cluster, and

$$A = \begin{pmatrix} x_1 - x_m & y_1 - y_m \\ \vdots & \vdots \\ x_n - x_m & y_n - y_m \end{pmatrix}$$

Then the 2×2 matrix $M = A^T A$ is:

$$M = \begin{pmatrix} \sum_i x_i^2 - x_m^2 & \sum_i x_i y_i - x_m y_m \\ \sum_i x_i y_i - x_m y_m & \sum_i y_i^2 - y_m^2 \end{pmatrix}$$

and contains the second moments of the points about their mean. M is symmetric and has real eigenvectors which are easily calculated and define the major and minor axes of an ellipse. The sides of the rectangle are determined by requiring that all points be contained in it.

The measure of goodness of fit is the ratio of the number of flagged points to the total number of coarse grid points enclosed by the rectangle. If the ratio is too small, the cluster is then processed into subclusters using a more sophisticated routine. We do not describe this method, since we have found the nearest-neighbor algorithm to perform sufficiently well for elliptic problems.

Before closing this section, we mention that this grid-generation technique can be easily extended to three dimensions. The nearest-neighbor algorithm would be unchanged. M would become a 3×3 matrix describing an ellipsoid.

2.6 Boundary Values for Refined Grids

Special care needs to be taken in specifying boundary values for the subgrid boundaries that are interior to the problem domain. Fine-grid boundary values are generally interpolated from the coarse-grid solution; accuracy must be maintained at the internal grid boundary. For time-dependent problems, the interpolated values must not destroy the stability of the time advancement. If the differential equation represents a conservation law, it is also desirable to maintain conservation at the boundary. We discuss how each of these concerns was addressed by Berger's method.

In one dimension, the boundaries of the fine grids are made to coincide with coarse-grid points. When both coarse- and fine-grid solutions have been integrated to the same time, there is no ambiguity in the choice of fine-grid boundary conditions. However, the fine grid also requires boundary conditions at intermediate times, since it is

advanced R times for each advancement on the coarse grid. Berger used linear interpolation in time, whose accuracy is consistent with the time-difference method. Higher-order interpolation would have required the storage of solution values from previous time steps.

In two dimensions, the interpolation was bilinear in space and linear in time. This method was found to provide sufficient accuracy, in part because internal boundaries were normally located where the solution was slowly varying.

Berger analyzed the stability of the time interpolation. With the Lax-Wendroff difference method applied to the linear wave equation in one space dimension, it was shown that these boundary conditions were stable. Numerical experiments with other hyperbolic equations in one and two dimensions showed no loss of stability.

For hyperbolic conservation laws in one dimension, it is well known that difference schemes that exactly conserve fluxes of the dependent variables can be guaranteed to converge to the correct shock speed and jump condition. Therefore, conservative methods are commonly used in shock calculations. When an internal grid boundary is introduced in the domain, special treatment must be applied to grid points along the boundary in order to maintain conservation, especially if a shock is located in the vicinity of the boundary. To preserve conservation, the flux into the grid boundary should exactly equal the flux out of it. Conservation can be imposed in two ways. Either the boundary conditions are specially evaluated, or the difference equations for the boundary points are modified to preserve the correct flux balance.

Berger derived conservative and stable boundary-difference formulas for use with the one-dimensional wave equation. It is not clear whether these conditions were used; however, it was not critical for the calculations shown, since shocks were always located away from internal grid boundaries.

Analysis was not done for two dimensions; due to the rotation, coarse and fine grid points will not coincide. Berger points out that spatial, bilinear interpolation is not conservative. However, her results indicate that accuracy was maintained.

Finally, we note that it would be difficult to implement modified boundary-difference equations for arbitrarily rotated grids. Conservative difference equations using fine and coarse grid points would be difficult to construct, given the arbitrary rotation. It is more straightforward to interpolate the boundary values in a conservative manner.

2.7 Data Structures

Because of the complexity of Berger's system of grids, non-standard data structures were used to describe the grid hierarchy and to store grid-solution vectors. We describe these structures in this section.

The purpose of the grid data structure is to describe each grid and its relationship to other grids. For example, the information stored in the data structure determines which coarse grid to interpolate boundary conditions for a fine grid, the order of integration of the grids, where the solution vector for a given grid is located, etc. The grids are organized in a tree-like structure. An illustration of a two-dimensional tree structure is given in Fig. 2.5. The tree is constructed by keeping track of the local relationships that are indicated by the double arrows in the figure.

Before describing the tree, we introduce some definitions. Subgrids of the same parent coarse grid are called siblings. Subgrids are called neighbors if they are at the same level of refinement but have different parents.

Each grid is treated as a node in the tree; its description is stored as a fixed-length vector. To distinguish them, the nodes are numbered. They contain a complete description of the grid; information stored includes:

1. Grid location (coordinates of its corners).
2. Number of grid points.
3. Mesh sizes.
4. Level in tree.
5. Parent.
6. Offspring.

7. Previous neighbor on level.
8. Next neighbor on level.
9. Time to which grid has been integrated.
10. Index in main storage array where solution is stored.

A parent can have more than one offspring, and a subgrid can have more than one parent (for example, $G_{3,1}$ in Fig. 2.5). Since node vectors contain a fixed number of elements, a linked list is used for data storage if a grid has multiple relatives. In that case, the node entry for the parent or offspring points to the beginning of the string of relatives that is located in the linked list. A description of the linked list and its implementation can be found in any standard computer science text on data structures.

This grid structure makes feasible an otherwise unwieldy data-management problem and is partly responsible for the overall efficiency of the adaptive method. The structure is dynamic in that grids can be added or deleted from the tree during program execution.

The storage of grid-solution vectors also must be dynamic. Grids created during the adaptive process require a location to store solutions. Also, the area occupied by solutions for grids that are destroyed should be reclaimed for future use. If this is not done, the program's memory requirements could exceed the computer's limit during a run. FORTRAN does not permit dynamic dimensioning of arrays, so a special data structure to handle the solution vectors was developed.

A single, large, global array is allocated for storing all solutions and any temporary work space that is required. When a grid is created, space is reserved in this array for its solution. The beginning address in the array is saved in the node vector. When a grid is destroyed, its space is reclaimed. Because of the dynamic nature of the creation and destruction of grids, storage in the global-solution array will not be contiguous. The location and length of free blocks in the array is maintained in a linked list. When a grid is created, the list is searched to find an appropriate location to store the solution. When a grid is deleted, its solution storage area is added to the list of free blocks.

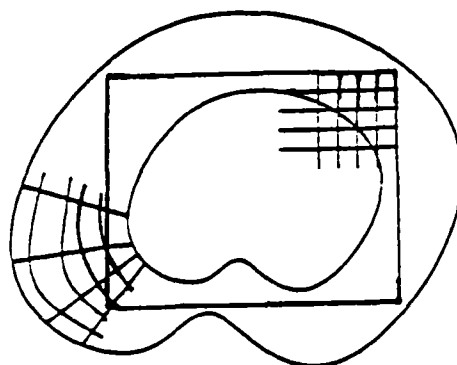


Fig. 2.1. Overlapping, component grids.

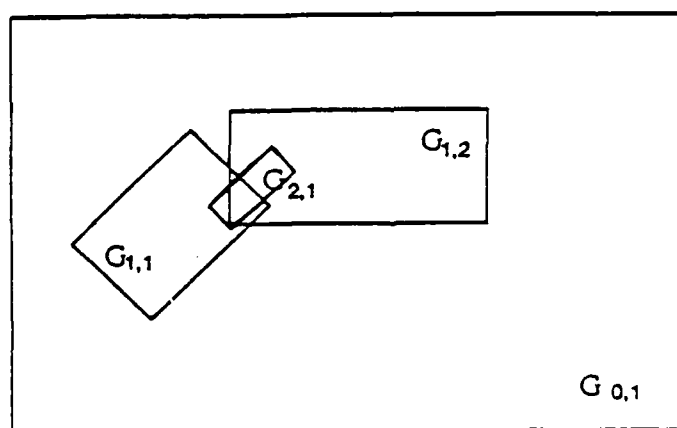


Fig. 2.2. Example of overlaid grid structure.

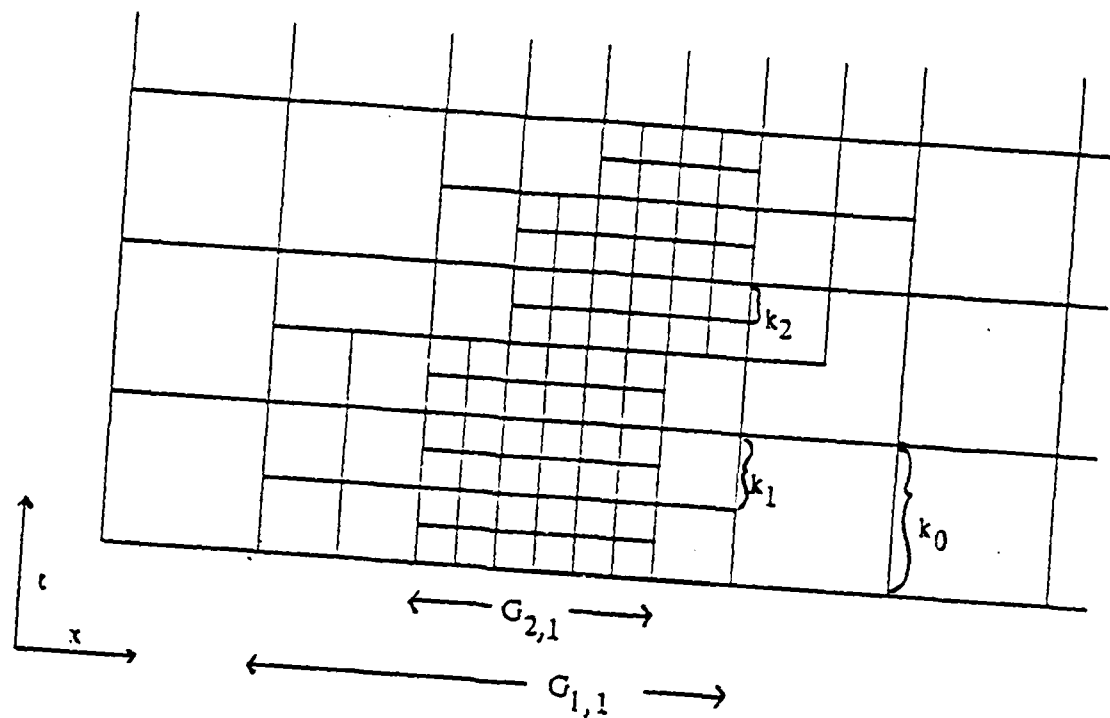


Fig. 2.3. Time integration on three-level grid structure.

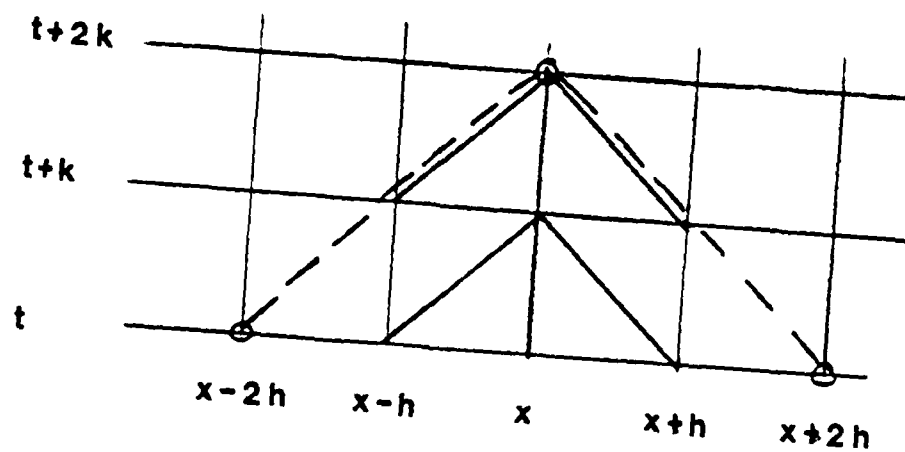


Fig. 2.4. Richardson extrapolation for time-explicit method.

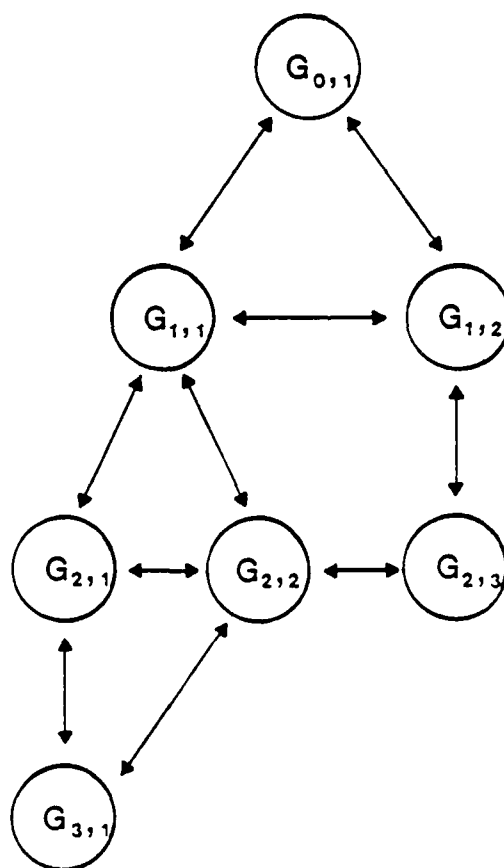
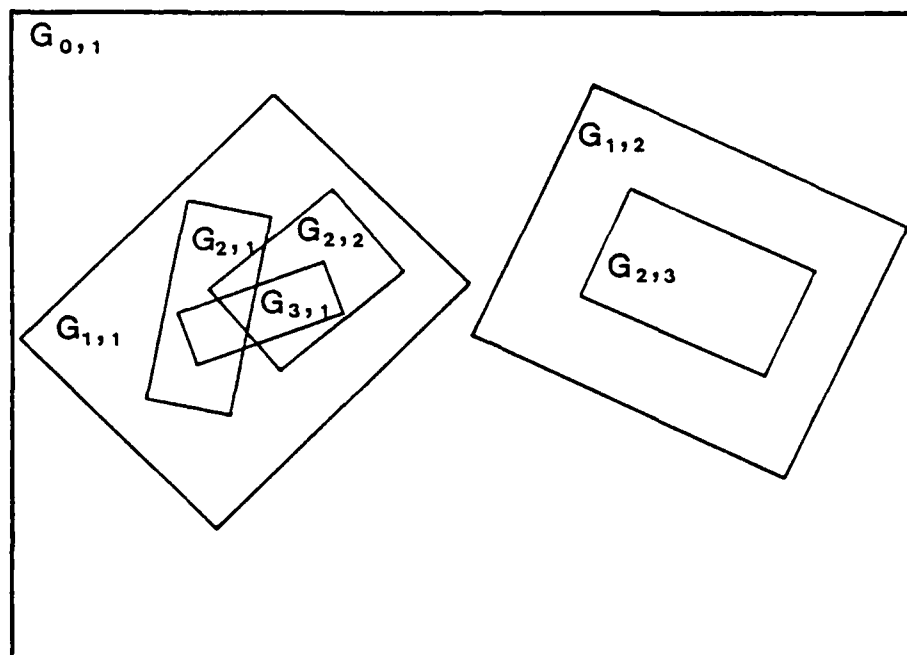


Fig. 2.5. Tree data structure

Chapter 3

STRATEGY FOR ELLIPTIC EQUATIONS

3.1 Overview

In this chapter, we describe how Berger's refinement approach is applied to elliptic flow problems. The differences prevent direct application of Berger's method. Our strategy was to implement the following two features of the method:

- overlaid, locally uniform grid refinement consisting of rotated rectangles, and
- refinement regions defined by Richardson error estimates.

Thus, Berger's grid-generation algorithms and data structures, described in Chapter 2, are used in our program.

In the next section, we discuss how the characteristics of elliptic equations influence the choice of the method. The differences between elliptic and hyperbolic flows are highlighted. We then develop two adaptive strategies, active and passive. Discussions of the error-estimation methods and treatment of initial and boundary conditions for refined grids follow.

3.2 Influence of Elliptic Equations on Adaptive Approach

The flows we are interested in are governed by the steady, incompressible, laminar or Reynolds-averaged Navier-Stokes equations. These systems are elliptic in contrast to the Euler equations, which are hyperbolic, and the boundary-layer equations, which are parabolic.

Elliptic flows are characterized by thin regions (boundary and free-shear layers), in which the velocity has a rapid variation in the direction normal to the main flow direction. This contrasts with hyperbolic flows which contain shocks, in which the rapid variation occurs over infinitesimally small regions aligned with or oblique to the main flow direction. Within shear layers, viscosity or turbulent diffusion is important.

Outside the thin shear layers (TSLs), the velocity varies smoothly, the effects of viscosity are small, and the flow is determined primarily by a balance between inertia and the pressure gradient. Viscosity can often be completely neglected in these regions.

Fine grids are needed in the vicinity of TSLs; coarser grids suffice for the outer flow. The locations and sizes of TSLs are usually not known in advance, and may depend on the Reynolds number. An adaptive method must be able to "recognize" the TSLs so that the refinement grids can be correctly located.

Elliptic flows may also have recirculating regions, in which the local flow direction is opposite to the main flow. Due to recirculation and the long-range effects of pressure, influences may be felt both upstream and downstream. In hyperbolic and parabolic flows, influence occurs only in the downstream direction.

Because information propagates differently, different solution methods are used for each type of flow. For hyperbolic and parabolic flows, the solution is generated by marching in the flow direction. With elliptic equations, large regions of the flowfield require simultaneous solution. This requires modification of the adaptive method used for hyperbolic equations.

3.3 Two Adaptive Strategies

In this section, we develop two methods; each is applicable to a class of elliptic flows. Overlaid grid refinement is used in both cases; they differ primarily in how the solutions on the various grids relate. Also, each uses a different error criterion. The classification of flows is discussed first. We then outline the two adaptive strategies and their solution and error-estimation procedures. The methods are described in detail in later sections.

When the shear layers are thin, the outer flow depends only weakly on the properties of the shear layers. The flowfield in each region can be studied independently to a good approximation (Mehta, 1984). Such a flow is described as having a weak viscous-inviscid interaction.

When shear layers are not thin, there is stronger coupling between the inner and outer flows. Local effects within the shear layer may then have long-range, multidirectional influence. The coupling is provided by the pressure gradient and recirculation. These flows are said to have strong viscous-inviscid interaction; the inner and outer regions cannot be analyzed independently.

Consider a concrete example — incompressible, viscous flow over an airfoil at a high Reynolds number. So long as the boundary layer remains attached, it is thin and exerts a weak influence on the outer flow. The latter can be accurately calculated by either neglecting the boundary layer or using a crude approximation to it. Given an accurate outer flow, the boundary layer can be calculated to a good approximation. However, if the boundary layer separates, it can thicken considerably and there may be significant modification of the outer flow. In this case, the two parts of the flow field must be calculated simultaneously.

Different solution-adaptive strategies were developed for these two types of interactions.

For flows with weak interactions, the outer solution does not need to be recalculated, as the accuracy of the calculation of the shear layer is improved through grid refinement. The "passive" adaptive method is formulated to take advantage of this.

In the passive scheme, an initial calculation is made on a coarse grid and its error is estimated. Refined grids are then constructed so that they enclose the inaccurate region(s). Care is taken to ensure that their boundaries be in accurate regions, so that accurate boundary conditions can be taken from the initial solution.

Solutions are then calculated on the refined grid regions, and their errors are estimated. If necessary, a new level of refined grids is created. The procedure is repeated until further refinement is no longer necessary.

At any step, the solution is calculated only on the most recently created, finest level grids. The outer solution is never recalculated; it assumes a "passive" role in the adaptive process. Since the outer

solution is not recalculated, its error, rather than the truncation error, must be used as the basis for a refinement criterion. This is an outline of the passive scheme; details will be given later.

For flows with strong interactions, the inner and outer solutions have to be calculated simultaneously. Improving the solution in the shear layer changes the flow in the outer region. The "active" scheme is formulated for these flows.

In the active scheme, an initial solution is calculated on a coarse grid, as in the passive method. Since the initial grid is coarse, the strong interaction causes the outer solution to be inaccurate. The solution is likely to be inaccurate everywhere, so the solution error is not a good indicator of where refinement is required. For this reason, the local truncation error is used as the basis for the criterion for refinement in the active scheme.

Refined grids are constructed to enclose regions of large truncation error. The coarse-grid solution provides boundary conditions for these grids, and the solution is calculated on them.

However, the coarse grid solution has to be recalculated to account for the change in the inner solution. This is done by modifying the coarse-grid equations to account for the improvement provided by the fine grid, then recalculating the coarse-grid solution. New fine-grid boundary conditions are obtained from the new coarse-grid solution, and the fine-grid solution recalculated. This procedure is repeated until the solution no longer changes on either grid. In this method, the coarse grid "actively" participates in the solution procedure.

The choice of which strategy to use, active or passive, depends on the strength of the coupling between the flow within the TSLs and the outer flow. For simple flows, the strength can be determined.

For flows along solid surfaces, the coupling is due to the displacement of the outer flow by the boundary layer. The strength of the interaction can be quantified. This was done for the developing boundary layer in a plane channel (shown in Fig. 3.1). We outline the analysis here; details are given in Appendix A.

The velocity V_0 in the inviscid core region can be related to the displacement thickness, δ^* . Let the change in δ^* be:

$$\Delta\delta^* = (K-1)\delta^*$$

where K is a sensitivity parameter. On a coarse grid, the boundary layer may be smeared to twice its actual size, $K = 2$. Figure 3.2 shows the relative change in V_0 as a function of δ^*/L for $K = 2$ and 10, where L is the channel width.

The relative change in V_0 should be no greater than the maximum allowable error. For $K = 2$ and maximum allowed relative error, 0.1%, the displacement thickness should be $\delta^*/L < 10^{-3}$. At larger δ^*/L , the coupling is stronger. The passive method should be used for $\delta^*/L < 10^{-3}$; the active method is required for larger δ^*/L .

This concludes the outline of our adaptive strategies. To summarize, we have shown that elliptic flows having strong and weak viscous-inviscid interactions require different adaptive strategies, and we have developed these strategies.

In the next two sections, we describe each strategy in more detail.

3.4 Passive Method

3.4.1 Strategy

In this section, we describe the algorithm for the passive strategy, which is applicable to flows having weak viscous-inviscid interactions. We also present some theoretical results that justify the technique.

The passive method is begun with the calculation of a converged solution on the coarse grid. The error in this solution is then estimated, using the method described below in Section 3.7. Regions having large estimated solution error are fit with refined grids, as described in Section 2.5. Boundary conditions and initial guesses for refined grids are interpolated from the coarse-grid solution. The solution is then calculated on all newly created refined regions (see Fig. 3.3). A special procedure is required to calculate the solution on a set of overlapping grids; it is described in Section 3.6.

The solution is not recalculated outside the refined regions in this method. Boundary values for the refined region remain fixed during the computation. For safety, each newly created rectangle contains a buffer zone, which is usually one or two coarse mesh widths wide on all sides.

After the solution and its error estimate have been computed, refined grids are generated. These grids usually lie within the boundaries of the previous grids, but they are not required to do so. Boundary conditions are taken from the next coarsest grid on which data are available.

Thus, the passive solution method is described by:

- (0) Set up base grid, G_0 ; $k = 0$.
- (1) Calculate solution on G_k .
- (2) Estimate solution error on G_k .
- (3) Check for convergence.
- (4) If converged, stop; otherwise continue.
- (5) Generate refined grids, G_{k+1} .
- (6) $k = k + 1$; go to (1).

Convergence is achieved when the estimated solution error (maximum absolute value or rms) falls below a specified value.

3.4.2 Summary of Theoretical Results

Theoretical justification for this solution technique can be found in analyses of one-dimensional boundary-value problems. We shall summarize these results.

The solution to the two-point boundary-value problem for the ordinary differential equation:

$$\begin{aligned}
 -\epsilon y'' + p(x)y' + q(x)y &= f(x) \\
 y(0) &= y_0 ; \quad y(1) = y_1 \\
 p(x) &> 0 ; \quad q(x) \geq 0
 \end{aligned}
 \tag{3.4.1}$$

where $\epsilon > 0$ is a small parameter, has a boundary layer of thickness $O(\epsilon)$ at $x = 1$.

The second-order central difference approximations are:

$$y'_j = \frac{y(x_{j+1}) - y(x_{j-1}))}{h} \quad (3.4.2)$$

$$y''_j = \frac{y(x_{j+1}) - 2y(x_j) + y(x_{j-1}))}{h^2}$$

where h is the mesh size, $x_j = (j-1)h$, and $1 \leq j \leq N$.

Numerical solution of problem (3.4.1) using the approximations (3.4.2) can be grossly inaccurate everywhere if the boundary layer is not resolved, i.e., if $\epsilon < h$.

As an example, consider (3.4.1) with $a = f = 0$, $p = 1$, $u(0) = 0$, and $u(1) = 1$. The exact solution is:

$$y(x) = \frac{1 - \exp(-x/\epsilon)}{1 - \exp(-1/\epsilon)} \quad (3.4.3)$$

Dorr (1970) showed that, for fixed h and even N , a central difference approximation has the solution:

$$\lim_{\epsilon \rightarrow 0} y(x_j, \epsilon) = \begin{cases} \frac{1}{N+1} & ; \quad j \text{ even} \\ -\infty & ; \quad j \text{ odd} \end{cases} \quad (3.4.4)$$

One-sided (upwind) differencing approximates the first derivative by:

$$y'_j = \frac{y(x_j) - y(x_{j-1}))}{h} \quad (3.4.5)$$

Using upwind differences for the first derivative but retaining second-order differences for the second derivative gives the solution (Il'in, 1969):

$$y(x_j) = \frac{1 - \left(\frac{\epsilon}{\epsilon+h}\right)^j}{1 - \left(\frac{\epsilon}{\epsilon+h}\right)^N} \quad (3.4.6)$$

The solution (3.4.6) behaves like the exact solution (3.4.3) away from $x = 1$. However, the boundary layer is several mesh widths thick. Thus, it is smeared unless $h < \epsilon$, i.e., unless the grid is fine enough.

Kellogg and Tsan (1978) derive error bounds for solving (3.4.1) using upwind differencing. Their results show that the error in the boundary layer does not pollute the solution away from $x = 1$. For our adaptive method, this means (at least for this problem) that we do not have to re-solve the problem on the coarse grid as the boundary layer is refined, if upwind differences are used. Our experience shows that the technique works in both one and two dimensions (see Chapters 4 & 5).

For $p < 0$, (3.4.5) must be replaced by:

$$y'_j = \frac{y(x_{j+1}) - y(x_j)}{h} \quad (3.4.7)$$

3.5 Active Method

The active technique is used for problems with strong viscous-inviscid interactions. We describe the active algorithm in this section.

In this strategy, the grid-refinement process is similar to the one used in the passive strategy, with two major differences. First, the solution is iterated over all refinement levels after a new level of grids has been added. Second, the refinement is based on local truncation error estimates rather than solution error estimates (cf. Section 3.3). The method used to estimate the local truncation error is given in Section 3.7.

Because the coarse solution is updated in the active method, we do not need to be as conservative in refining the grid. Thus, the error tolerances used to define the refinement regions may be larger, and buffer zones for refined grids may be smaller.

Consequently, an active calculation generates smaller refined grids than a passive calculation of the same problem. However, the active calculation may not be more efficient, since additional work is expended in updating the outer solution. The relative efficiency of the two methods depends on the strength of the coupling between outer and inner

solutions. The passive method is better for weak coupling, the active method for strong coupling. It is difficult to determine a priori which method is more appropriate for a problem with an intermediate-strength coupling; it is safer to use the active strategy in such cases.

We shall describe the active method for generating the solution on the two-level system shown in Fig. 3.4. Extension to more levels is straightforward.

Assume we have the differential equation (written in operator form):

$$Lu = f \quad (3.5.1)$$

on the domain, Ω , with appropriate boundary conditions. L is the differential operator.

We want to solve (3.5.1) using the two-level grid system indicated in Fig. 3.4. The coarse grid G_0 covers Ω_0 and has mesh size H . The fine grid G_1 covers Ω_1 and its mesh size is h . Note that $H = Rh$, where R is the refinement ratio.

We use the same difference approximation on both grids; only the mesh size differs. The approximation on the coarse grid is represented as

$$L_H u_H = f \quad (3.5.2)$$

and on the fine grid as:

$$L_h u_h = f \quad (3.5.3)$$

We want the solution to satisfy the fine grid approximation on Ω_1 , the coarse grid approximation on $\Omega_0 - \Omega_1$, and the solutions on the two grids should agree at common points, i.e., on Ω_1 .

In other words, the solution should satisfy:

$$L_h u_h = f \quad \text{on } \Omega_1 \quad (3.5.4a)$$

$$L_H u_H = f \quad \text{on } \Omega_0 - \Omega_1 \quad (3.5.4b)$$

$$u_H = u_h \quad \text{on } \Omega_1 \quad (3.5.4c)$$

The last condition requires that the coarse grid equations be modified in Ω_1 . Equations (3.5.4b) are solved on $\Omega_0 - \Omega_1$, while the equations:

$$L_H u_H = L_H u_h \quad (3.5.5)$$

are solved on Ω_1 . The terms on the right-hand-side are the "coarse grid corrections terms". Solution of this equation gives the desired result (3.5.4c).

Assuming that the grids have been selected, the following procedure is used to generate the solution. An initial coarse grid solution is first calculated on the entire domain, i.e., $L_H u_H = f$ is solved on Ω_0 . Boundary conditions for the fine grid are interpolated from this solution, and a fine-grid solution is calculated. The coarse grid correction terms are then calculated, using the current fine grid solution. The modified coarse grid equations are solved, boundary conditions are then interpolated for the fine grid, the solution calculated on the fine grid, and so on. Convergence is achieved when the interpolated fine grid boundary conditions no longer change.

Thus, the active solution algorithm is:

- (0) Calculate initial coarse grid solution; solve:

$$L_H u_H^0 = f \quad (\text{on } \Omega_0) ; \quad m = 1$$

- (1) Update boundary values for G_1 by interpolating solution on G_0 :

$$u_h^m(\gamma_1) = u_H^{m-1}(\gamma_1)$$

- (2) Check for convergence, e.g.,

$$\max_{\gamma_1} |u_h^m(\gamma_1) - u_h^{m-1}(\gamma_1)| < \varepsilon$$

- (3) If converged, stop; otherwise continue.

- (4) Calculate fine grid solution; solve:

$$L_h u_h^m = f \quad (\text{on } \Omega_1)$$

- (5) Calculate coarse grid correction:

$$L_H u_h^m$$

(6) Calculate coarse grid solution; solve:

$$L_H u_H^m = \begin{cases} L_H u_h^m & \text{on } \Omega_1 \\ f & \text{on } \Omega_0 - \Omega_1 \end{cases}$$

(7) Go to (1).

Corrections are applied at coarse grid points internal to the fine grid. Coarse-grid points lying on the fine grid boundaries are not corrected, permitting the solution on the fine grid boundaries to change as the solution converges.

For one-dimensional problems, calculation of the correction is simple, since coarse and fine grid points are coincident. With rotated grids in two dimensions, the fine grid solution has to be interpolated to evaluate the correction terms. This is done as follows. The coarse-grid difference approximation (3.5.2) at a point (i,j) on a uniform rectangular coarse grid, G_0 , can be written:

$$\begin{aligned} L_H u_H \Big|_{ij} = & a_{ij}^H(u_H)_{ij} + b_{ij}^H(u_H)_{i+1,j} + c_{ij}^H(u_H)_{i-1,j} \\ & + d_{ij}^H(u_H)_{ij+1} + e_{ij}^H(u_H)_{ij-1} \end{aligned} \quad (3.5.6)$$

for the five-point stencil shown in Fig. 3.5. We have assumed that the equation is linear and the coefficients a, b, c, d, e are known, for the purposes of illustration. (The procedure is equally applicable to nonlinear equations.) The correction terms in (3.5.6) are formed by replacing the u_H 's in the stencil by the corresponding fine grid values, i.e.:

$$\begin{aligned} L_H u_h \Big|_{ij} = & a_{ij}^H(u_h)_{ij} + b_{ij}^H(u_h)_{i+1,j} + c_{ij}^H(u_h)_{i-1,j} \\ & + d_{ij}^H(u_h)_{ij+1} + e_{ij}^H(u_h)_{ij-1} \end{aligned} \quad (3.5.7)$$

The location of the coarse grid point (i,j) determines which solution values (coarse or fine) are used. Four cases are indicated in Fig. 3.5.

In case (i), all five coarse grid points are internal to the fine grid, G_1 ; the fine grid solution is used at all five points. In the second case, $(i+1, j)$ and $(i, j-1)$ lie outside the fine grid. The coarse grid values are used for these, the fine grid, for the other three. In case (iii), the point (i, j) lies on the boundary of G_1 . The correction term is not needed here nor for case (iv), for which the stencil is completely outside the fine grid domain.

This concludes our discussion of the active solution method for the two-level grid system; the method is easily generalized to more than two levels. Correction terms are applied on grid G_k , if there is a finer grid G_{k+1} . However, the solutions on the various grids are generated successively, going from coarsest to finest and back again. One iteration is defined as solving on the grids in the order:

$$G_1, G_2, \dots, G_{k_{\max}}, G_{k_{\max}-1}, \dots, G_0$$

where $G_{k_{\max}}$ is the grid on the finest refinement level. This "V" sweep pattern is also used in multigrid methods.

To summarize, the major difference from the passive method is that, in the active method, information is passed from the fine grid to the coarse grid and the solution is generated on both grids simultaneously. Also, the truncation rather than the solution error is used to define the refinement regions.

3.6 Solution on Overlapping Grids

On a given level, Berger's method may generate overlapping grids to enclose a cluster of flagged points. An example having two such grids is shown in Fig. 3.6.

Assuming that boundary conditions are specified, except in the overlap region, the problem is to solve a boundary-value problem on an irregular domain. A numerical version of the Schwarz alternating method, outlined below for the case of two rectangles, is used; it can be extended to more rectangles.

The problem domain, Ω , is composed of two overlapping rectangular subdomains, Ω_1 and Ω_2 , as indicated in Fig. 3.6. Γ_1 and Γ_2

denote the boundaries of the two rectangles; γ_1 and γ_2 are their internal boundaries.

Boundary conditions are specified on the external boundaries, $(\Gamma_i - \gamma_i)$, $i = 1, 2$. The initial conditions on the internal boundaries, γ_1 , are guessed (or interpolated from a coarse grid solution) and the following algorithm is carried out.

- (1) Update internal boundary values on γ_1 from the solution on Γ_2 : $u_1^n(\gamma_1) = u_2^{n-1}(\gamma_2)$. This usually requires interpolation.
- (2) Solve for $u_1^n(x, y)$ (on Ω_1).
- (3) Update internal boundary values on γ_2 : $u_2^n(\gamma_2) = u_1^n(\gamma_1)$, again by means of interpolation.
- (4) Solve for $u_2^n(x, y)$ (on Ω_2).
- (5) Check for convergence, e.g.,

$$\max_{\gamma_i} |u_i^n(\gamma_i) - u_i^{n-1}(\gamma_i)| < \epsilon ; \quad i = 1, 2$$

- (6) If converged, stop; otherwise go to (1).

The Schwarz alternating method has been analyzed for simple elliptic equations (including Laplace's and Poisson's) on unions of simple subdomains. Some of the relevant results are summarized next.

The technique was originally developed to prove the existence of continuous solutions to the Laplace-Dirichlet problem on irregular regions. (See Stoutemyer (1972) for a review of the early literature.) However, assuming the existence of a continuous solution, the convergence of the discrete solution can be guaranteed for certain classes of problems. Miller (1965) shows that two additional conditions are sufficient for convergence. They include:

- (1) the discrete approximations must converge to the continuous solution as the mesh size goes to zero;
- (2) subregion approximations must satisfy the maximum principle on their respective domains (i.e., the solution error on each subregion must be less than the error on the internal boundaries. This condition is satisfied by many elliptic equations.)

Miller showed that, at the n^{th} iteration, the error on the internal boundaries converges linearly:

$$||e_i^n||_{\gamma_i} \leq q^n ||e_i^0||_{\gamma_i} ; \quad i = 1, 2$$

where $||\cdot||_{\gamma_i}$ denotes the maximum value along the boundary γ_i , and $|q| < 1$. Convergence can be accelerated using standard techniques, e.g., Aitken's method (see Smith, 1978), SOR (successive overrelaxation), etc. The magnitude of q is problem-dependent; it depends on the amount of overlap, the geometry of the subdomains, etc.

Miller also analyzed the case in which the numerical solution on each grid is not carried to convergence at each step. He found bounds on the error made on the internal boundaries in terms of the error at internal points and thus established stability. However, the convergence rate could not be determined, since it depends on the numerical method and the degree of solution convergence on each subregion. All of Miller's results apply to both linear and nonlinear problems.

Rodrigue and Simon (1983) analyzed a method for solving elliptic equations on multiprocessor computers. They decomposed the domain into subregions, each of which was assigned a single processor. The Schwarz alternating procedure was used to compute the solution on the complete domain. The method was recast as a matrix problem so that classical techniques of acceleration could be applied (see e.g., Hageman and Young, 1981). For linear problems, they showed that the procedure is equivalent to the block Gauss-Seidel iterative method. Numerical experiments with Laplace's equation showed that the convergence rate increased with increased overlap area.

Tang et al. (1985) analyzed the Schwarz method for Poisson's equation in n -dimensions when the problem domain is a line, rectangle, or box in 1-D, 2-D, and 3-D, respectively. The domain was composed into an arbitrary number of strips, each having the same size and amount of overlap. The convergence rate was linear, but depended on the overlap area and the number of strips. SOR applied to the internal boundary values was shown to speed up convergence. Formulas were given for the optimum relaxation factors. Numerical results agreed with the theory.

Kang et al. (1985) consider the Schwarz alternating procedure with multiple subregions on parallel computers. They demonstrated convergence independent of the order of solution on the subregions. They also showed that an energy norm of the error monotonically decreases.

A practical application of the Schwarz method is given in Atta and Vadyak (1983). Transonic external flow calculations were made using overlapping, three-dimensional grids. Two overlapping component grids were used. Multivariate quadratic interpolation was used to update internal boundary values. Starting from an initial specified field, each grid was iterated a fixed number of times before boundary values were transferred. Convergence was achieved after 10-15 Schwarz iterations.

3.7 Error Estimation

In the passive technique, we need to determine where the coarse grid solution is accurate. We keep that part of the solution, and refine and re-solve only where the error is large.

In strongly coupled problems, the solution, or global, error is spread over the domain by convection and diffusion. Consequently, it may not be a good indicator of where grid refinement is required. Therefore, in the active method, the local truncation error is used to define refinement regions.

In this section we present the methods used to estimate the solution and truncation errors. We then discuss how the two errors interact in elliptic flows.

We use a form of Richardson extrapolation to estimate both types of error. This technique assumes that the solution error can be expressed as a Taylor series

$$e(h,x) = u(0,x) - u(h,x) = h^p F(x) + h^q G(x) + \dots \quad (3.7.1)$$

where $u(0,x)$ is the exact solution, h the mesh size, and p the order of the method ($p = 2$ for second-order methods). This expansion is valid for smooth solutions with several continuous derivatives -- a condition that is satisfied by all elliptic flows.

If we double the mesh size (for both coordinates in two dimensions) and calculate another solution, the error becomes:

$$\begin{aligned} e(2h, x) &= u(0, x) - u(2h, x) \\ &= 2^p h^p F(x) + 2^q h^q G(x) + \dots \end{aligned} \quad (3.7.2)$$

Subtracting (3.7.2) from (3.7.1) and dividing by $2^p - 1$ gives an estimate of the solution error:

$$\begin{aligned} \tilde{e}(h, x) &= \frac{u(h, x) - u(2h, x)}{2^p - 1} \\ &= h^p F(x) + h^q \left(\frac{2^q - 1}{2^p - 1} \right) G(x) + \dots \end{aligned} \quad (3.7.3a)$$

Comparing (3.7.1) with (3.7.3a), we see that:

$$\tilde{e}(h, x) = e(h, x) + O(h^q) \quad (3.7.3b)$$

so the estimate is accurate to order q .

Next we derive the truncation error estimate. In operator form, the difference method is:

$$L_h[u(h, x)] = f \quad (3.7.4)$$

where L_h is the difference operator. The truncation error is defined as the residual obtained by substituting the exact solution of the differential equation, $u(0, x)$, into the difference equation, i.e.,:

$$\begin{aligned} \tau(h, x) &= L_h[u(0, x)] - f \\ &= L_h[u(0, x)] - L_h[u(h, x)] \end{aligned} \quad (3.7.5)$$

If L_h is linear, the relationship between solution and truncation errors is obtained by combining the definitions (3.7.1) and (3.7.5):

$$\tau(h, x) = L_h[e(h, x)] \quad (3.7.6)$$

Consequently, for linear problems, the solution error estimate, (3.7.3a), can be substituted into (3.7.6) to yield an estimate of the truncation error:

$$\tilde{\tau}(h, x) = L_h[\tilde{e}(h, x)] \quad (3.7.7)$$

The accuracy of this estimate is found by inserting (3.7.3b) into (3.7.7):

$$\begin{aligned}\tilde{\tau}(h,x) &= L_h[e(h,x) + O(h^q)] \\ &= \tau(h,x) + L_h[O(h^q)]\end{aligned}\tag{3.7.8a}$$

Since L_h is a difference approximation to an r^{th} order differential operator, it multiplies function values by terms as large as h^{-r} . Therefore:

$$\tilde{\tau}(h,x) = \tau(h,x) + O(h^{q-r})\tag{3.7.8b}$$

so that this estimate is accurate to order $(q-r)$. Comparing this with (3.7.3b), we see that solution error estimates are more accurate than truncation error estimates.

For nonlinear problems, we use (3.7.3a) to compute:

$$\tilde{u}(0,x) = u(h,x) + \tilde{e}(h,x)\tag{3.7.9}$$

which is used in place of the exact solution $u(0,x)$ in (3.7.5) to give the truncation error estimate:

$$\tilde{\tau}(h,x) = L_h[\tilde{u}(0,x)] - f\tag{3.7.10}$$

In practice, to estimate the error, the mesh size is doubled in both directions and a solution is calculated on the $2h$ grid for use in (3.7.3). For the passive method, the error is estimated only on the finest level at each adaptive step. For the active method, all grid meshes are doubled, and an active solution is calculated on the $2h$ grid system.

Before closing this section, we further discuss how the solution and truncation errors interact. We first derive a formal relationship and then give a phenomenological interpretation.

Consider the linear, elliptic differential equation:

$$Lu = f\tag{3.7.11}$$

and its solution:

$$u = L^{-1}f\tag{3.7.12}$$

where L is the differential operator and L^{-1} represents its Green's function.

Providing L_h can be inverted, from (3.7.4) we can also write:

$$u(h,x) = L_h^{-1}[f] \quad (3.7.13)$$

Noting the similarity between (3.7.12) and (3.7.13), L_h^{-1} can be interpreted as a discrete Green's function (Gladwell and Wait, 1979). We can also invert (3.7.6):

$$e(h,x) = L_h^{-1}[\tau(h,x)] \quad (3.7.14)$$

The numerical approximation (3.7.4) introduces the truncation error τ ; (3.7.14) indicates how τ is converted into the resulting solution error. Note that the solution error is reduced by reducing the truncation error, i.e., by refining the grid.

We give an interpretation of how the error gets distributed in elliptic flows. Consider the linear problem:

$$u(x,y) \frac{\partial \phi}{\partial x} + v(x,y) \frac{\partial \phi}{\partial y} = \epsilon \nabla^2 \phi + S_\phi \quad (3.7.15)$$

on a regular domain with Dirichlet boundary conditions. This equation describes the convection and diffusion of a passive scalar ϕ in a known velocity field, given by u and v . S_ϕ is a source of ϕ in the field.

A numerical approximation for this problem is:

$$u \frac{\delta \phi_h}{\delta x} + v \frac{\delta \phi_h}{\delta y} = \epsilon \nabla_h^2 \phi_h + S_\phi \quad (3.7.16)$$

where $\delta/\delta x$, $\delta/\delta y$, ∇_h^2 are difference operators. This can also be written:

$$L_h \phi_h = S_\phi \quad (3.7.17)$$

The solution error, $e_h = \phi - \phi_h$ is related to the truncation error by (3.7.6), which when expanded becomes:

$$u \frac{\delta e_h}{\delta x} + v \frac{\delta e_h}{\delta y} = \epsilon \nabla_h^2 e_h + \tau_h \quad (3.7.18)$$

We see that the solution error satisfies a discrete convection-diffusion equation similar to (3.7.16), with $e_h = 0$ on the boundaries. τ_h acts as a source term. This shows that the solution error is convected and diffused over the flow field in the same manner as ϕ , and that its source is the truncation error.

3.8. Initial Guesses and Boundary Values for Refined Grids

Since iterative solution methods are used, initial guesses are required for newly created grids and the grids used for the error estimates. For the former, initial guesses are interpolated from existing grids. Guesses for $2h$ -grids are interpolated from the corresponding h -grid solution. In both cases, bilinear interpolation is used.

The cell in the grid containing the point at which the guess is needed is first found, as indicated in Fig. 3.7. The interpolated solution is found in terms of the local cell coordinates:

$$\begin{aligned} \xi &= i - i_0 \\ \eta &= j - j_0 \end{aligned} \quad (3.8.1)$$

where (i_0, j_0) is the origin (see Fig. 3.7), and i and j are considered continuous variables. In these coordinates, bilinear interpolation is:

$$\begin{aligned} u(\xi, \eta) &= (1-\xi)(1-\eta) u_{i_0, j_0} + \xi(1-\eta) u_{i_0+1, j_0} \\ &+ \eta(1-\xi) u_{i_0+1, j_0+1} + \xi\eta u_{i_0, j_0+1} \end{aligned} \quad (3.8.2)$$

This second-order accurate method is sufficient for providing initial guesses.

Boundary conditions for refined grids are interpolated from the finest existing grids. These boundaries normally fall within the problem domain and are thus fictitious internal boundaries. Ideally, the order of accuracy for interpolation at these locations should be at least $r + p$, where r is the order of the differential equation and

p the order of accuracy of the approximation (Bai, 1984). This makes the accuracy at the fictitious boundaries consistent with the accuracy at other internal points.

To illustrate, assume we approximate the second-order equation ($r = 2$):

$$\frac{d^2 u}{dx^2} = f \quad (3.8.3)$$

with second-order central differencing ($p = 2$):

$$\frac{u_{j+1} - 2u_j + u_{j-1}}{h^2} = f + O(h^2) \quad (3.8.4)$$

where we have indicated the leading term in the truncation error. Note that u_{j+1} , u_j , and u_{j-1} are the exact solution values.

Also assume that we have to use an interpolated value \bar{u}_j at some point J inside the domain. \bar{u}_j differs from the exact value u_j by:

$$\bar{u}_j = u_j + O(h^q) \quad (3.8.5)$$

where q is the order of accuracy of the interpolation method. To see how the interpolation error affects the difference approximation at the point J , substitute (3.8.5) into (3.8.4) to get:

$$\frac{u_{j+1} - 2(\bar{u}_j + O(h^q)) + u_{j-1}}{h^2} = f + O(h^2) \quad (3.8.6)$$

Maintaining the same order of accuracy requires $q \geq 4$; cubic interpolation would be satisfactory.

We used bilinear interpolation (similar to Berger's method) for fictitious internal boundaries, primarily because of its simplicity. Our experience indicates that the accuracy was sufficient for the problems investigated (see Chapter 5). However, a more accurate method, preferably cubic interpolation, is recommended, since it will increase the adaptive method's efficiency.

Exact boundary values are used where refined grid boundaries are on the computational boundary.

For the Navier-Stokes equations, a conservative interpolation procedure was developed for internal fine grid boundaries. This method is discussed in detail in Chapter 7.

In this chapter we have described how Berger's method was extended to provide a method of solving the elliptic flow equations. New solution and error estimation procedures were presented, along with a discussion of the treatment of initial guesses and boundary values for refined grids. We apply the method to one-dimensional problems in the next chapter, and to a two-dimensional one in Chapter 5. Chapters 6 and 7, respectively, discuss the solution method and adaptive procedures used with the Navier-Stokes equations. Results of adaptive calculations for the Navier-Stokes equations follow in Chapter 8.

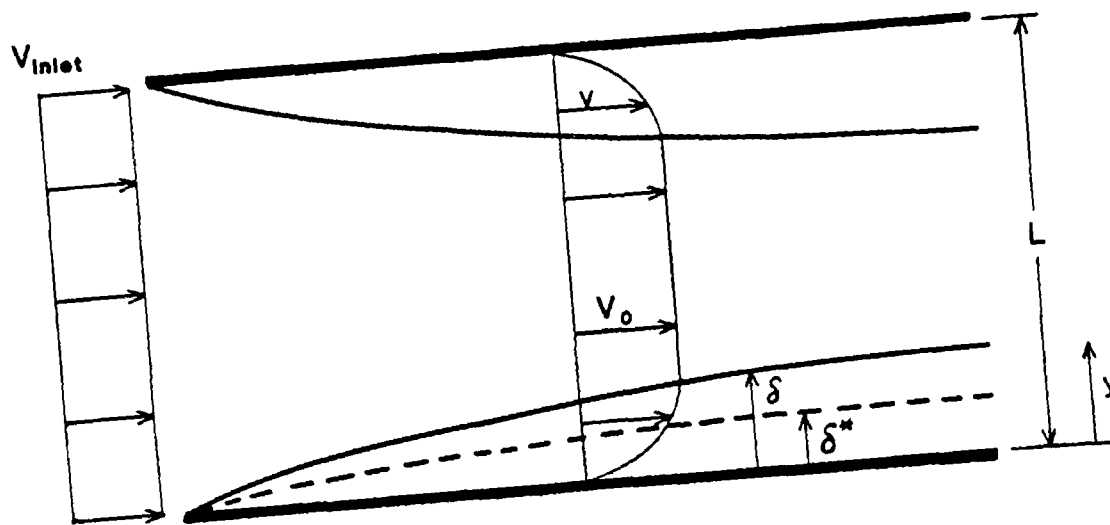


Fig. 3.1. Developing boundary layer in a plane channel.

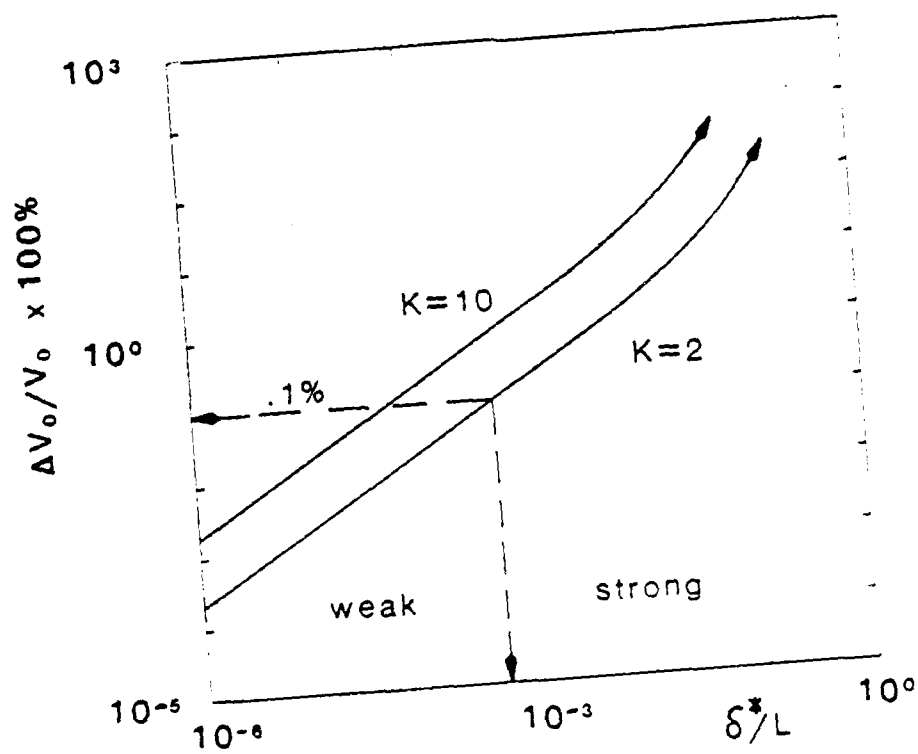


Fig. 3.2. Quantification of boundary-layer coupling.

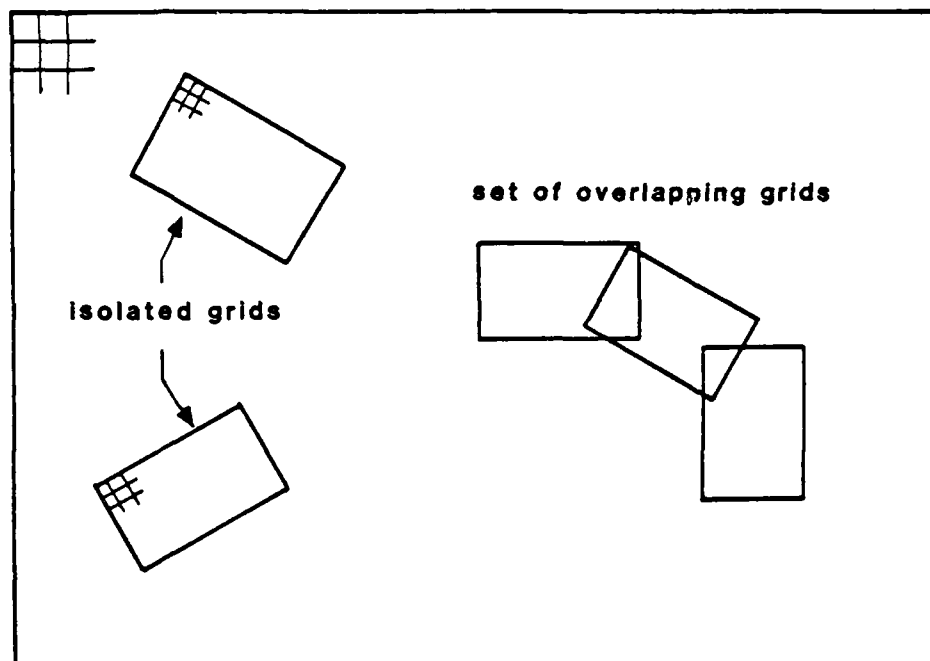


Fig. 3.3. Example of newly refined regions.

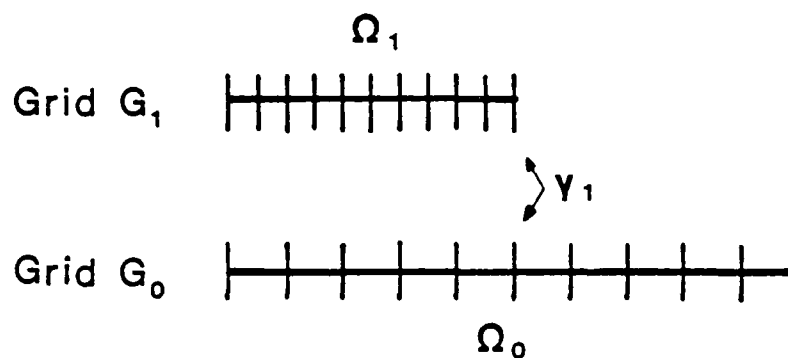


Fig. 3.4. Notation for active solution on two-level grid system.

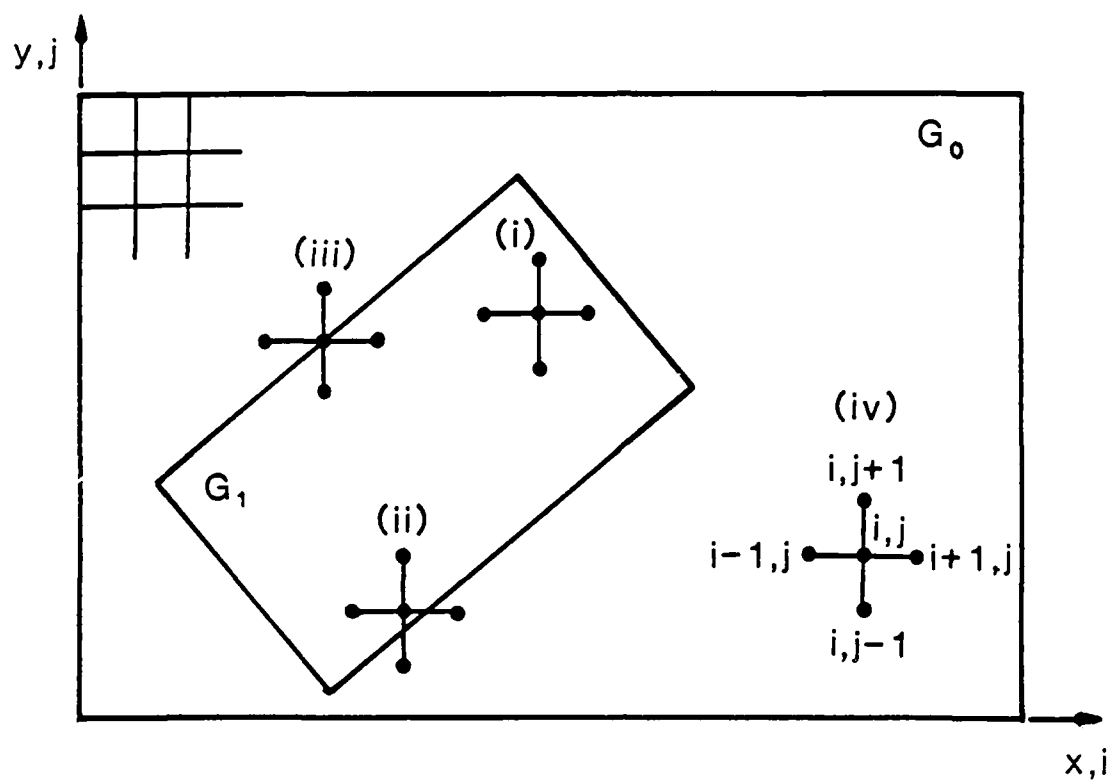


Fig. 3.5. Difference stencils and coarse-grid correction terms.

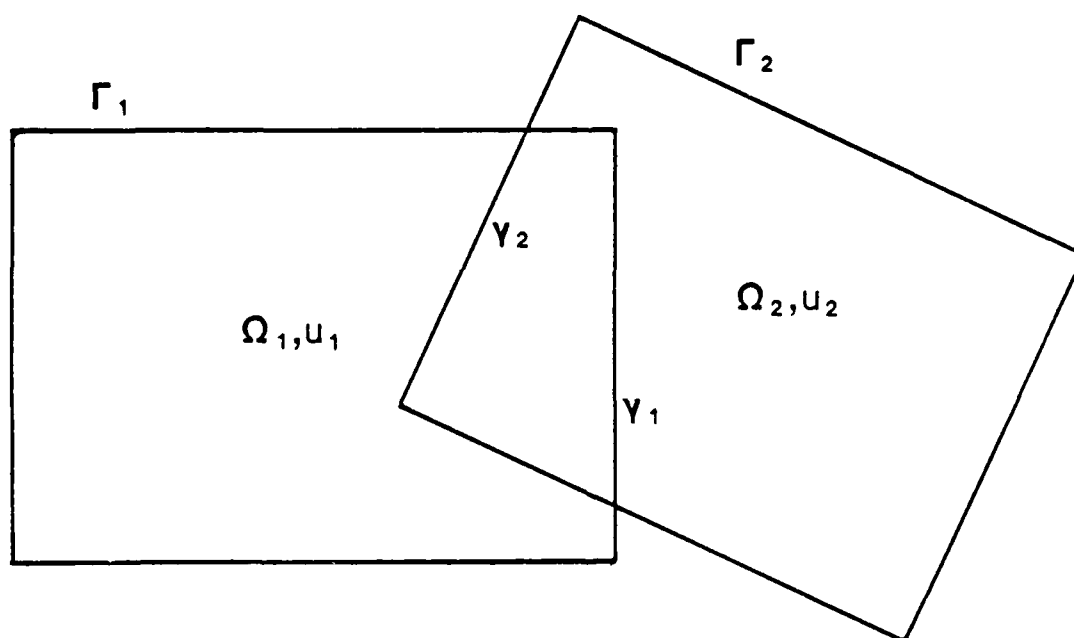


Fig. 3.6. Notation for solution on overlapping grids.

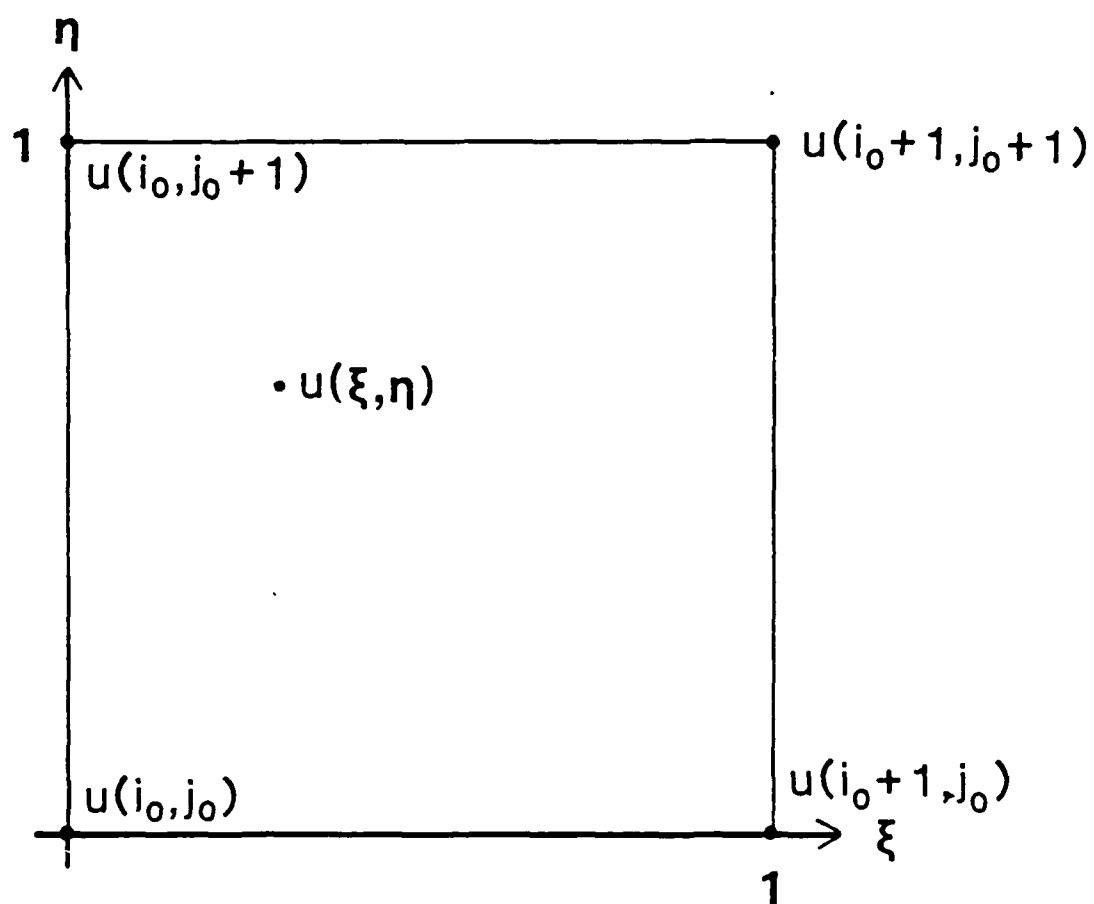


Fig. 3.7. Local cell coordinates for bilinear interpolation.

Chapter 4

APPLICATION TO ONE-DIMENSIONAL BOUNDARY-VALUE PROBLEMS

4.1 Introduction

In this chapter, we apply our method to one-dimensional boundary-value problems. The purpose is to demonstrate the feasibility and performance of the technique.

The passive adaptive method described in the previous chapter is applied to the linear, singular perturbation problem:

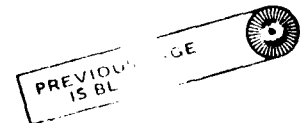
$$\epsilon y'' + a(x)y' + b(x)y = f(x) \quad (4.1)$$

$$y(x_1) = y_1 ; \quad y(x_2) = y_2$$

where $\epsilon > 0$ is a small parameter. Solutions of (4.1) have boundary and/or internal layers of thickness ϵ in which there is rapid solution variation. Outside these layers, the solution is smooth. In general, only boundary layers can exist if $a(x)$ does not change sign on the interval, $[x_1, x_2]$. If $a(x)$ changes sign, an internal layer can appear at the turning point, $a(x) = 0$. (See Bender and Orszag (1978) for further discussion of the theory of singular perturbation problems.)

Equation (4.1) is a common model problem used to test numerical methods for the Navier-Stokes equations. With $b(x) = 0$, ϵ is analogous to the viscosity, $a(x)$, to the velocity, and the source term, $f(x)$, to the pressure gradient.

The passive method is used for (4.1), since the inner and outer solutions are weakly coupled. For small ϵ , accurate outer solutions can be constructed analytically by neglecting the second derivative, dropping one boundary condition and integrating the resulting problem. The coupling is weak, because the outer solutions can be accurate even when the boundary layer is completely neglected.



4.2 Numerical Method

Recall that, in the passive method, the solution outside the refinement region is not improved as finer grids are added. Since the grid gets refined mainly in the boundary layers, we must use a numerical method that can give accurate results outside of boundary layers when a coarse mesh is used. As discussed in Section 3.4, only first-order upwind differencing has this characteristic; it is therefore used for approximating (4.1).

The first derivative is approximated by one-sided differences:

$$y'_j = \begin{cases} \frac{y(x_{j+1}) - y(x_j)}{h} & \text{if } a(x_j) \geq 0 \\ \frac{y(x_j) - y(x_{j-1}))}{h} & \text{if } a(x_j) < 0 \end{cases} \quad (4.2a)$$

and the second derivative by central differences:

$$y''_j = \frac{y(x_{j+1}) - 2y(x_j) + y(x_{j-1}))}{h^2} \quad (4.2b)$$

on a uniform grid, where $h = x_{j+1} - x_j$. The resulting system of difference equations is tridiagonal and is solved using the Thomas algorithm.

The method (4.2) is used for all grids; the boundary conditions differ for each.

4.3 Adaptive Procedure

An initial solution is calculated on a base grid. The mesh spacing on this grid is doubled and another solution calculated using the same method. The error is then estimated by inserting these two solutions in (3.7.3a) with $p = 1$. Points having errors larger than a prescribed tolerance δ_{\max} are flagged, clustered, and fit with refined grids. Boundary conditions for new grids are taken from the coarse grid. Solutions are then calculated on each refined grid using (4.2), the error estimated, and finer-level subgrids created. The process is repeated

until the maximum estimated error is less than δ_{\max} or a minimum mesh size is achieved on the finest-level grid(s).

The following rules are used to generate a refined grid from a collection of flagged points:

1. Adjacent flagged points belong to the same refined grid.
2. Flagged points separated by m or less consecutive unflagged points are in the same grid.
3. Grids are buffered with n unflagged points on their boundaries.

n and m are small integers that are user specified.

Steps 1 and 2 are the nearest-neighbor clustering algorithm in one dimension. The parameters n and m allow the user some control over the refined grid sizes; we have used values in the range $1 < n, m < 5$.

Note that, in 1-D, refined grid boundaries coincide with parent grid points; boundary values for fine grids are the parent grid values at these locations. The mesh size for a subgrid is obtained by dividing the parent's mesh size by a constant factor, R , the refinement ratio. An example grid structure after 1 and 2 refinement steps is shown in Fig. 4.1a & b, where $R = 2$ and $m = n = 1$.

4.4 Numerical Results

We illustrate the method by showing the results for a simple boundary layer. Results for more complex boundary layers follow.

Unless otherwise noted, all calculations were made with the following parameters:

1. Perturbation parameter, $\epsilon = 10^{-2}$.
2. Maximum allowable estimated error, $\delta_{\max} = 10^{-3}$.
3. Maximum number of consecutive unflagged points in a cluster, $m = 1$.
4. Buffer region equal to one parent mesh length, $n = 1$.
5. Refinement ratio, $R = 2$.

In all plots, adaptive results are indicated by circles, exact solutions (or very fine grid solutions) are plotted as solid lines.

Example 1 - Simple Boundary Layer

The first problem has a single boundary layer and is given by:

$$\epsilon y'' - y' = 0 \quad (4.3)$$

with boundary conditions:

$$y(-1) = 1 \quad (4.4a)$$

$$y(1) = 2 \quad (4.4b)$$

This problem has the exact solution:

$$y(x) = 1 + \frac{(e^{x/\epsilon} - e^{-1/\epsilon})}{2 \sinh(1/\epsilon)} \quad (4.5)$$

Figure 4.2a shows the initial solution calculated on a grid having 11 mesh points. The outer solution is accurate, even though there are no points in the boundary layer. For comparison, a solution calculated on the same grid using second-order central differencing for the first derivative is shown in Fig. 4.2e.

The error was estimated in the initial solution, and the criteria indicated the need for a grid spanning $0 \leq x \leq 1$. The boundary condition at $x = 0$ was taken from the coarse grid solution. The right boundary condition was (4.4b). The method (4.2) was used to calculate the solution on this grid; it is shown in Fig. 4.2b.

The error in this solution was estimated, and the third-level grid spanned $0.4 \leq x \leq 1$. The solution calculated on this grid is shown in Fig. 4.2c. As expected, the size of each successive refined grid decreases and the boundary layer is better resolved after each adaptation.

The complete adapted solution is given in Fig. 4.2d. The boundary layer is accurately resolved with an efficient placement of mesh points. Seven refinement levels were needed; Table 4.1 gives a description of all grids that were generated.

The efficiency of the method can be assessed by comparing its computational work with that of a uniform grid calculation with mesh size equal to the smallest mesh used in the adaptive method.

We estimate the work for these problems. In adaptive calculations, the majority of the work is consumed in inverting the tridiagonal systems. This work is proportional to the number of grid points. We multiply the cost of the solution on each grid by 1.5 to include the cost of error estimation and grid generation. For the uniform grid, we use only the actual work.

For the simple boundary layer, the uniform grid requires five times the adaptive grid work for these parameters. Note that this savings will increase geometrically for similar problems in two and three dimensions.

Example 2 - Two Boundary-Layer Problem

A problem having boundary layers at both endpoints is described by:

$$\begin{aligned}\epsilon y'' - 2xy' + 2y &= 0 \\ y(-1) &= y(1) = 1\end{aligned}\tag{4.6}$$

The results of an adaptive calculation are shown in Fig. 4.3. The solid line is a solution calculated using a fine uniform grid. Both boundary layers are accurately resolved, and the estimated adaptive work is two-thirds of the uniform grid work.

Example 3 - Internal Boundary-Layer Problem

A problem having an internal boundary layer due to a turning point is:

$$\begin{aligned}\epsilon y'' + xy' &= 0 \\ y(-1) &= 1 ; \quad y(1) = 2\end{aligned}\tag{4.7}$$

The results of the adaptive calculation are plotted in Fig. 4.4 along with a fine grid solution. Note that the two knees in the solution receive the most refinement. The region of steep gradient between the

knees is nearly linear and does not need much refinement. An error criterion based on solution gradient would concentrate points in this region. The adaptive calculation is estimated to be 25% faster than a uniform grid calculation.

Example 4 - Boundary-Layer Problem with Non-constant Outer Solution

The problem:

$$\begin{aligned}\epsilon y'' - y' &= -1 \\ y(0) &= y(1) = 0\end{aligned}\tag{4.8}$$

has a boundary layer at $X = 1$ with a linearly varying outer solution. For this problem, $\epsilon = 10^{-3}$. The uniform grid work is estimated to be three times the adaptive grid work.

The adaptive calculation is plotted versus a fine grid solution in Fig. 4.5. The boundary layer is accurately resolved, with most grid points placed at the sharp knee in the curve. Only 25% of the initial coarse grid had to be refined.

Example 5 - Problem with a Turning Point and a Boundary Layer

The last example has a boundary layer and a turning point. (This is one of several singular perturbation examples given in Pearson, 1968.) The differential equation is:

$$\epsilon y'' + |x|y' + (x - 1/2)^3 y = 0\tag{4.9}$$

with the boundary conditions:

$$y(-1) = 1 ; \quad y(1) = 2$$

For this problem, the maximum allowed estimated error $\delta_{\max} = 2.5 \times 10^{-2}$. The adaptive calculation is plotted versus a fine grid solution in Fig. 4.6. Four distinct refinement regions are created in this problem. Beginning at the left boundary, grid points are concentrated in the boundary layer and at the sharp knee in the curve there. Next follows a region of moderate grid density where the solution is

moderately varying. The grid-point density is also high at the knees associated with the internal layer. The estimated adaptive work is approximately 30% less than that for a uniform grid.

4.5 Conclusions

These examples demonstrate the feasibility of using the passive solution technique for solving singular perturbation problems. Use of the upwind difference approximation is crucial to the success of the method. Figure 4.2e indicates that, if central differencing had been used, the whole grid would have been refined.

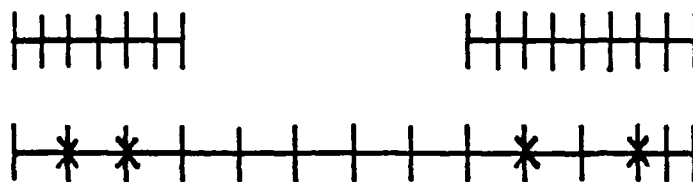
Estimates of the computational work indicate that the adaptive method is more efficient than a uniform grid, even allowing for the additional overhead (primarily error estimation for these problems).

No stability or convergence problems associated with adaptation of the grid were encountered. A smooth distribution of grid points was generated. Points were concentrated in regions of large curvature; steep, linear regions were not overrefined.

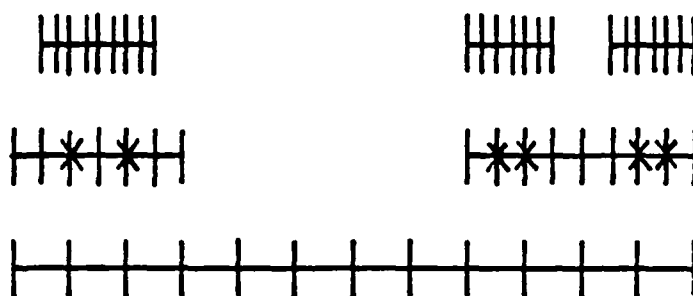
The Richardson-solution error estimates reliably indicated where the solution was accurate and where refinement was required, justifying the expense of computing the solutions used for the estimate.

TABLE 4.1
REFINED GRIDS FOR EXAMPLE 1

LEVEL	x_1 (LEFT BOUNDARY)	x_2 (RIGHT BOUNDARY)	NUMBER OF MESH POINTS	Δx
0	-1.00	1.00	11	0.2000
1	0.00	1.00	11	0.1000
2	0.30	1.00	15	0.0500
3	0.65	1.00	15	0.0250
4	0.78	1.00	19	0.0125
5	0.84	1.00	27	0.0063
6	0.89	1.00	37	0.0031
7	0.92	1.00	55	0.0016



(a) 2 level grid structure



(b) 3 level grid structure

Fig. 4.1. Example 1-D grid structures.

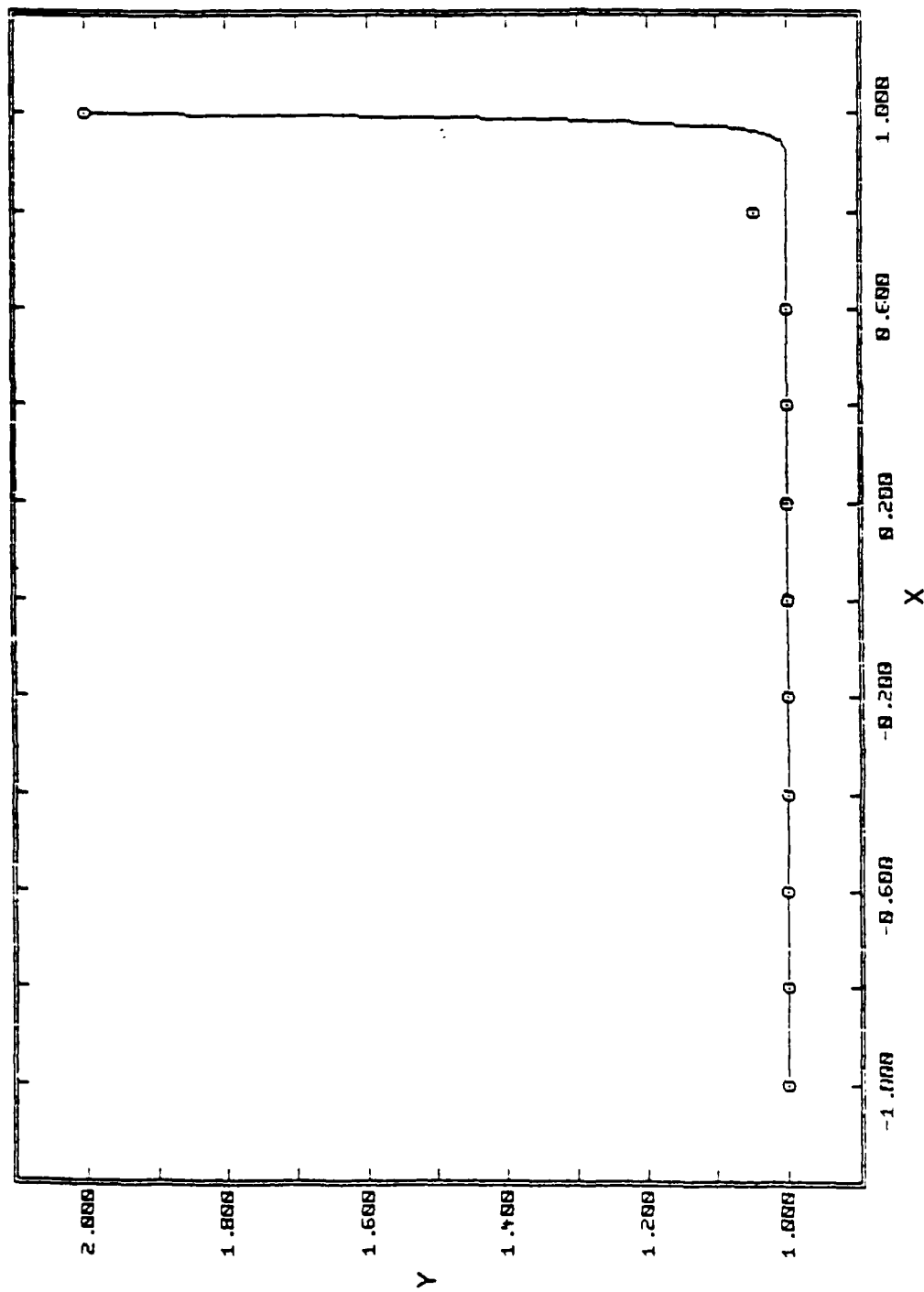


Fig. 4.2(a). Simple boundary layer: base grid solution.

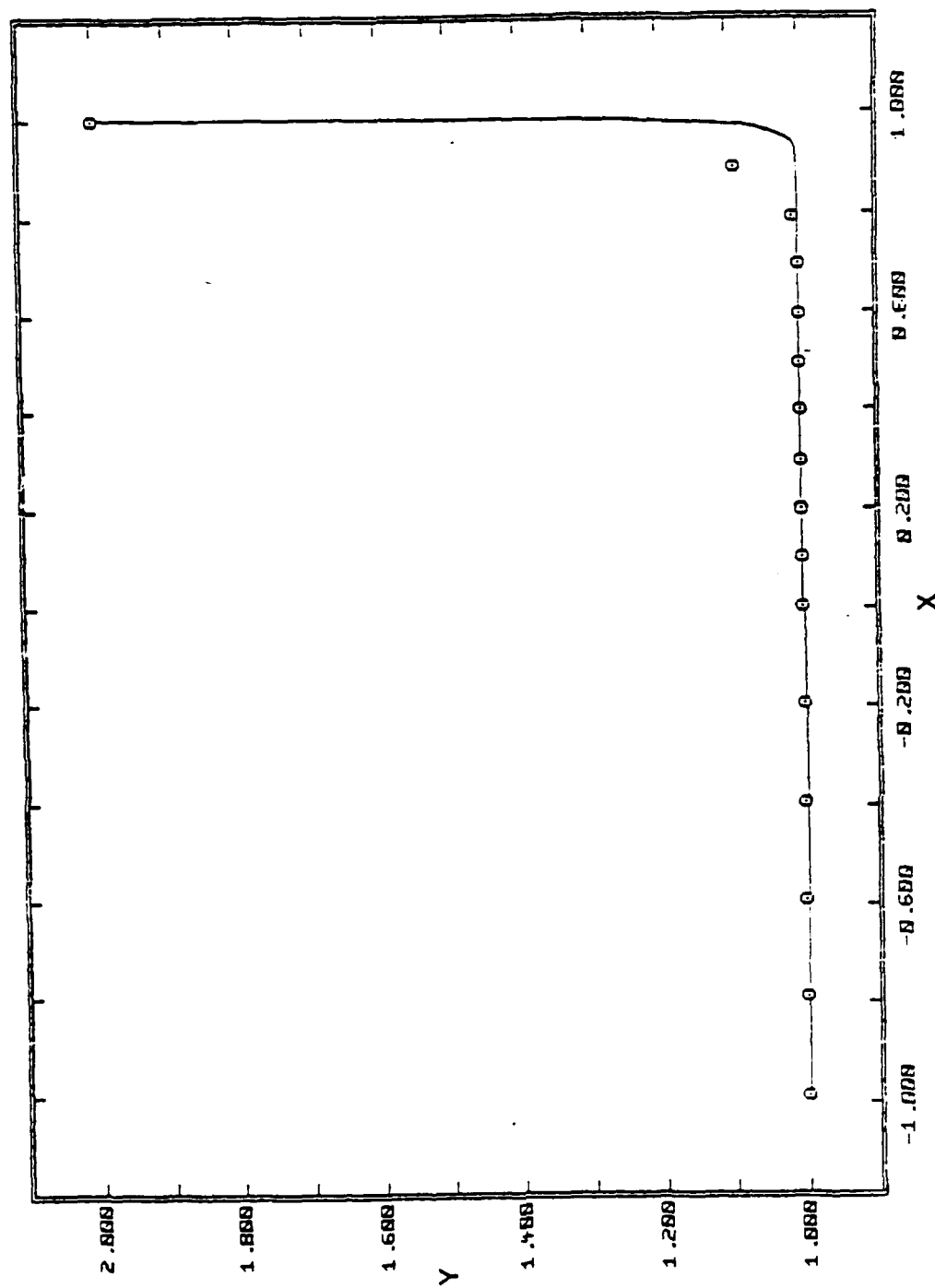


Fig. 4.2(b). Simple boundary layer: solution on second-level grid.

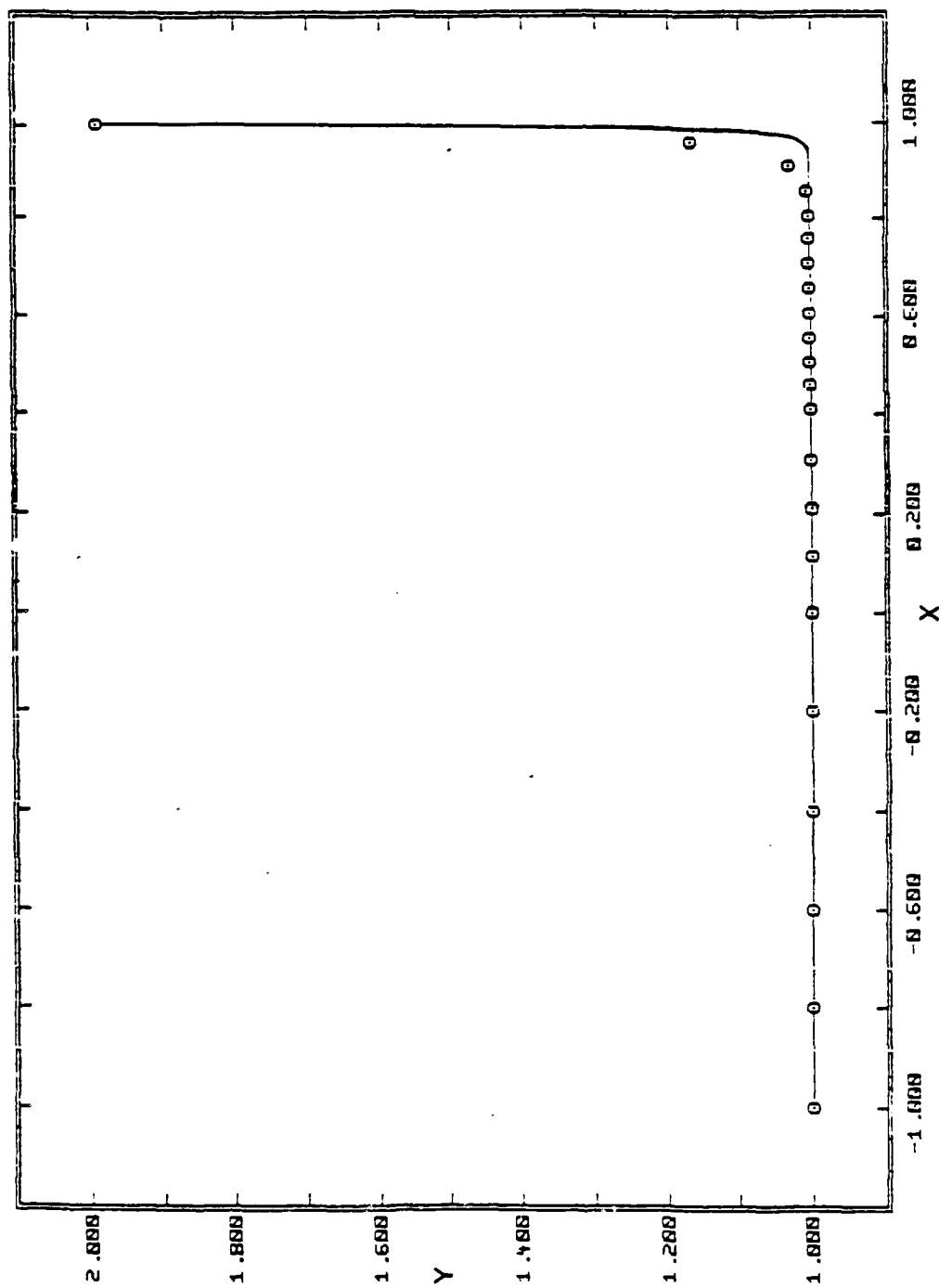


Fig. 4.2(c). Simple boundary layer: solution on third-level grid.

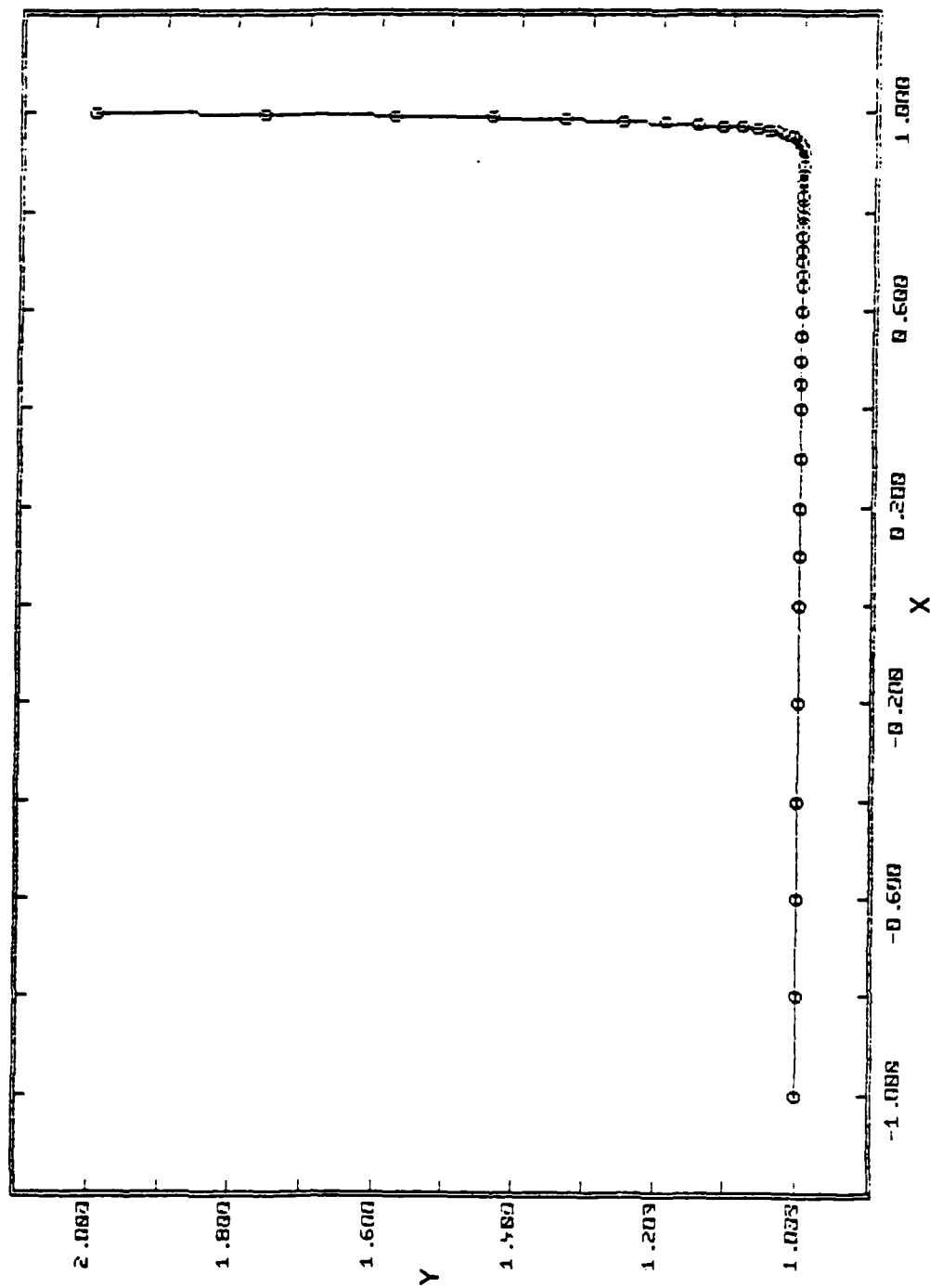


Fig. 4.2(d). Simple boundary layer: complete adapted solution.

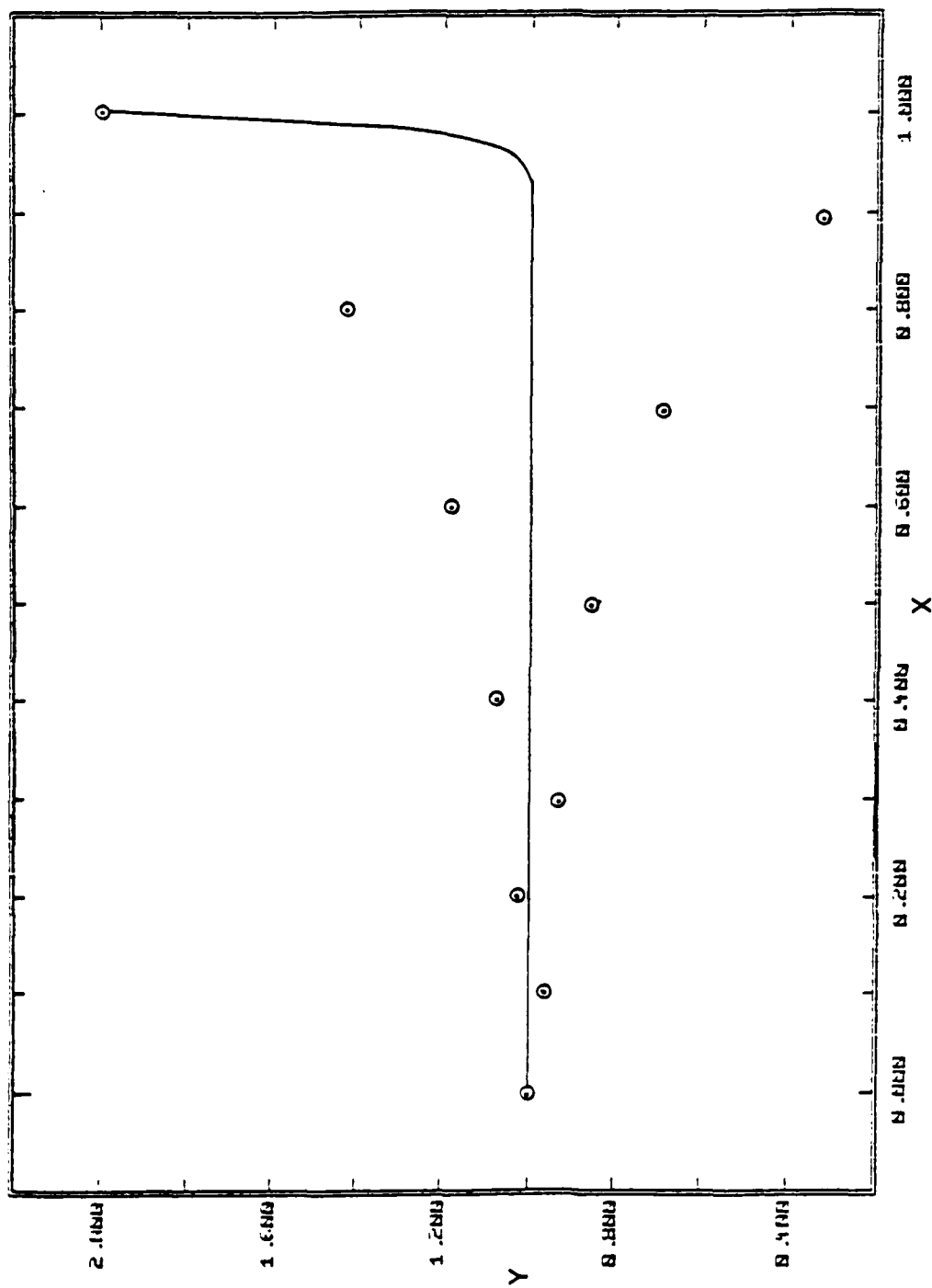


Fig. 4.2(e). Simple boundary layer: central difference solution on base grid.

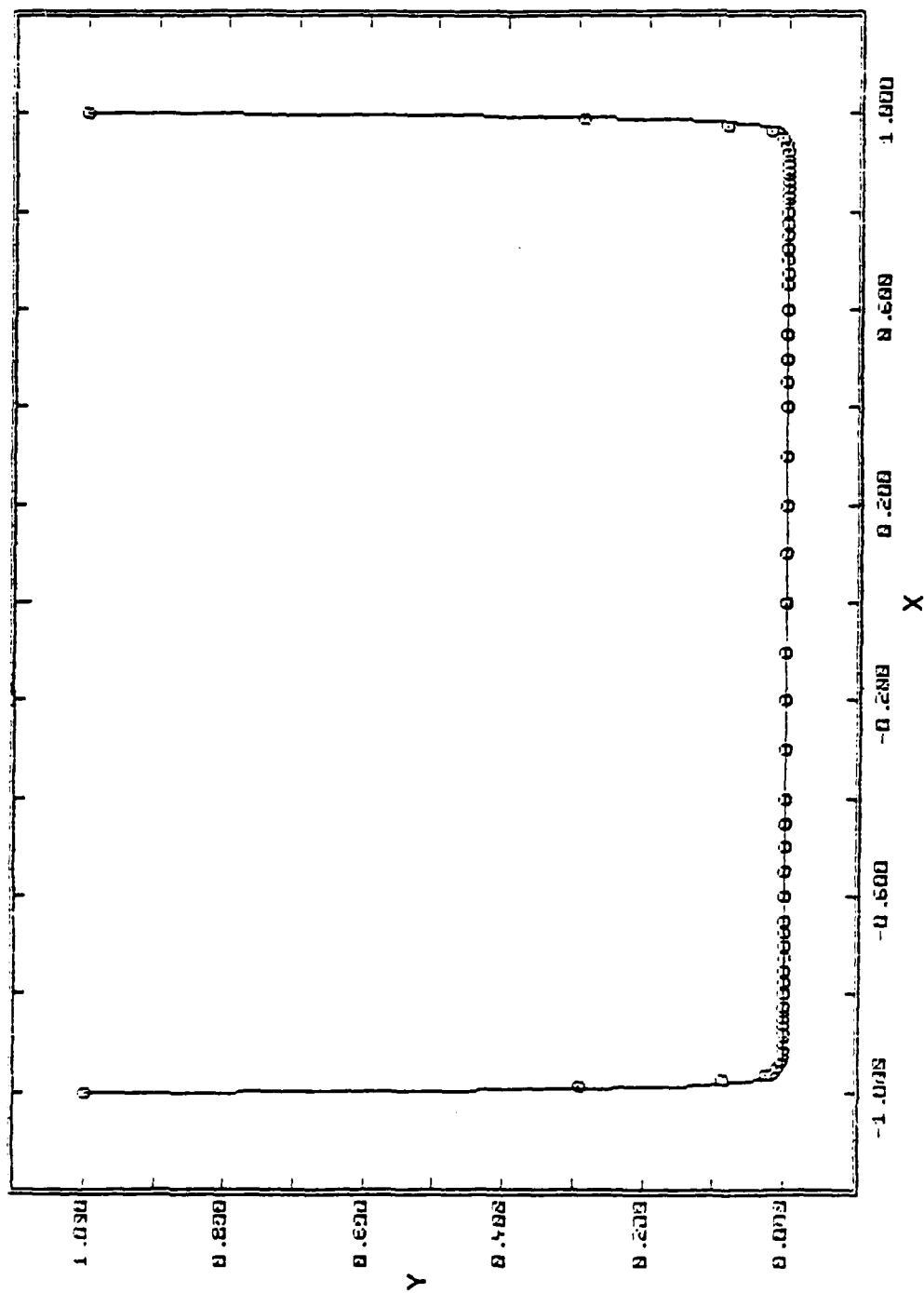


Fig. 4.3. Adapted solution for two-boundary-layer problem.

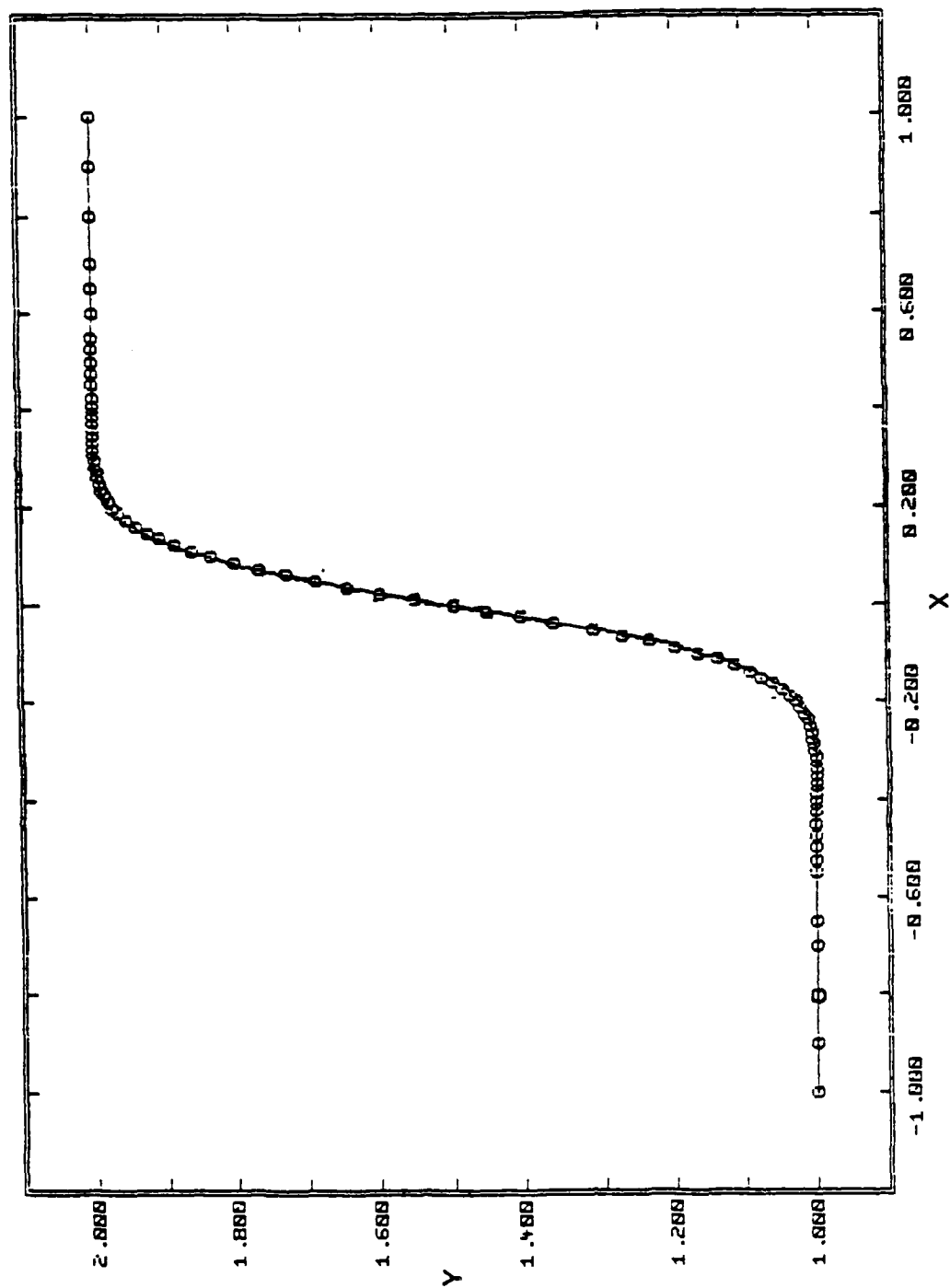


Fig. 4.4. Adapted solution for internal boundary-layer problem.

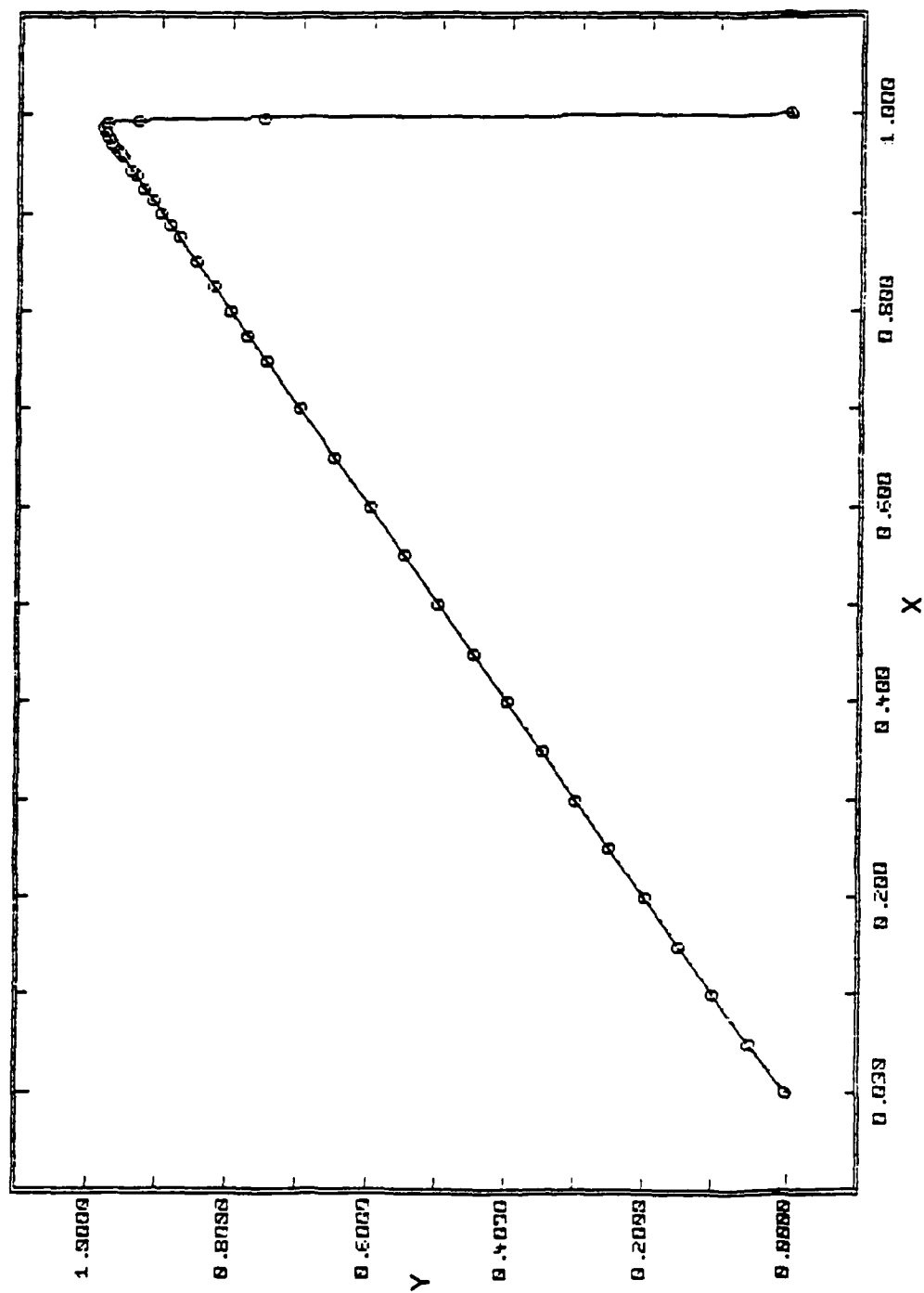


Fig. 4.5. Adapted solution for boundary layer with non-constant outer solution.

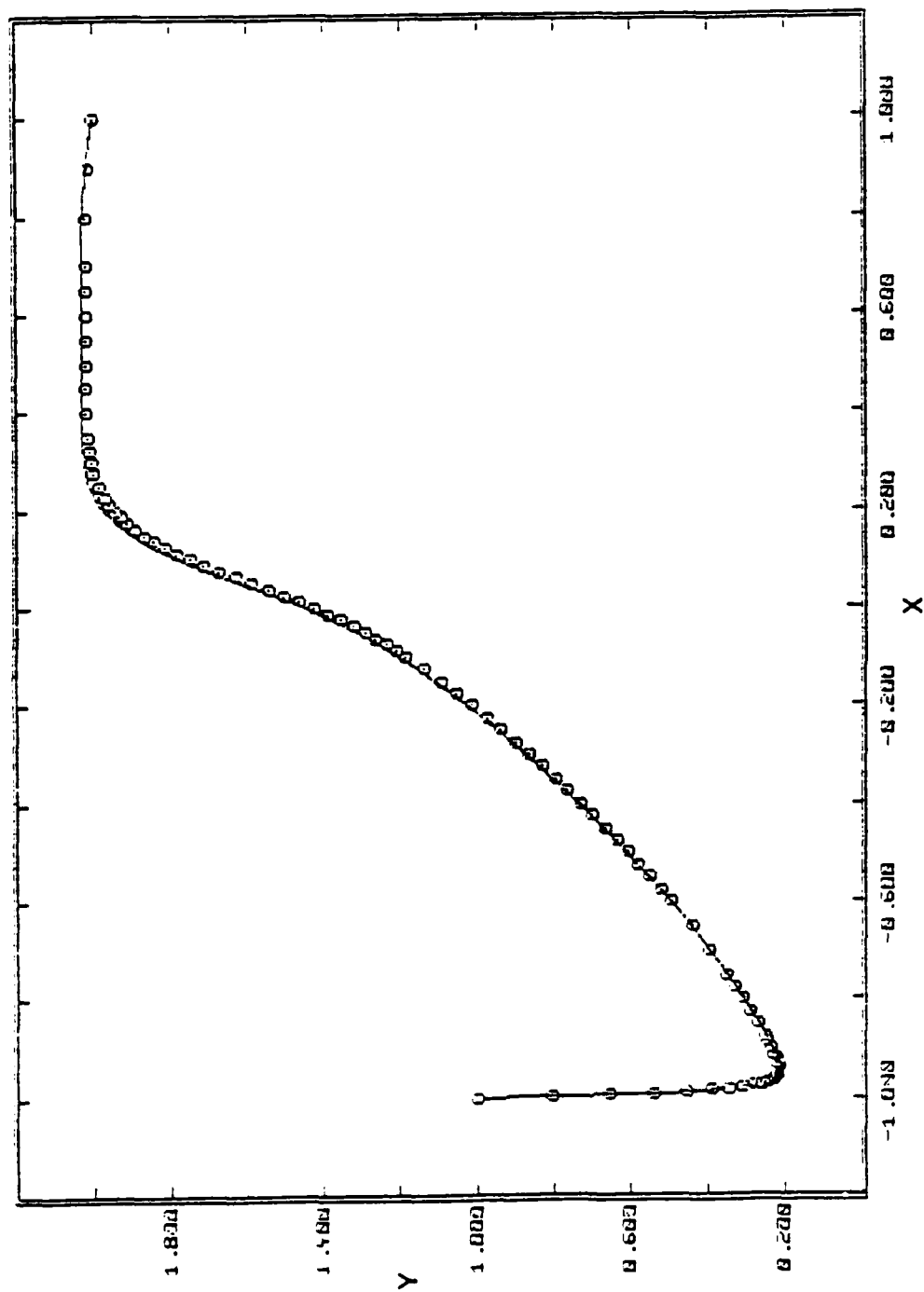


Fig. 4.6. Adapted solution for problem with a turning point and a boundary layer.

Chapter 5

APPLICATION TO A TWO-DIMENSIONAL, LINEAR, CONVECTION-DIFFUSION PROBLEM

5.1 Introduction

In this chapter, we apply our method to the two-dimensional, linear, convection-diffusion problem:

$$\begin{aligned} u\phi_x + v\phi_y &= \epsilon(\phi_{xx} + \phi_{yy}) \\ v &= (u^2 + v^2)^{1/2} \end{aligned} \tag{5.1.1}$$

This is another model problem for the Navier-Stokes equations. Because it is linear and elliptic, it is appropriate for testing our method.

With the addition of source terms, (5.1.1) describes the convection and diffusion of passive scalars in a flow field. Thus it is connected with convective heat and mass transfer, which are important in energy-conversion systems.

Since the problem is two-dimensional, numerical diffusion is present when the flow is oblique to the grid lines. This error arises because approximations to derivatives are obtained from one-dimensional Taylor expansions along coordinate lines. Special methods have been devised to take account of the local flow angle, e.g., Jameson's rotated difference scheme (Jameson, 1974) and skew-upwind differencing (Raithby, 1976). We expect our method to align the coordinate system in the refined regions with the flow; special difference procedures should not be required. Problem (5.1.1) is a useful test of the ability of the method to accomplish this.

The passive method was used for adaptive solutions of this problem; justification for using this method was gained from numerical experimentation. However, for a comparison, calculations were also made using the active method.

5.2 Numerical Method

Equation (5.1.1) is nondimensionalized using the magnitude of the velocity V and a reference length L to obtain:

$$u' \phi_{x'} + v' \phi_{y'} = Pe^{-1} (\phi_{x'x'} + \phi_{y'y'}) \quad (5.2.1)$$

where

$$x' = \frac{x}{L} ; \quad y' = \frac{y}{L} ; \quad u' = \frac{u}{V} ; \quad v' = \frac{v}{V}$$

The Peclet number Pe is given by:

$$Pe = \frac{VL}{\varepsilon}$$

To use rotated rectangles, we transform from (x,y) coordinates to arbitrary rotated coordinates (ξ,η) related by:

$$\begin{pmatrix} \xi \\ \eta \end{pmatrix} = A \begin{pmatrix} x' \\ y' \end{pmatrix} \quad (5.2.2)$$

where

$$A = \begin{pmatrix} \cos \theta & \sin \theta \\ -\sin \theta & \cos \theta \end{pmatrix}$$

and θ is the angle of rotation with respect to the base grid. Equation (5.2.1) transforms to:

$$\tilde{u} \phi_{\xi} + \tilde{v} \phi_{\eta} = Pe^{-1} (\phi_{\xi\xi} + \phi_{\eta\eta}) \quad (5.2.3)$$

where

$$\begin{pmatrix} \tilde{u} \\ \tilde{v} \end{pmatrix} = A \begin{pmatrix} u' \\ v' \end{pmatrix}$$

Thus the form of the equations is invariant to rotation of the coordinate system when the velocities are given with respect to the rotated coordinates.

First-order upwind differencing is again used for first derivatives, and second-order central differences for the second derivatives. The resulting difference equation is:

$$u^+ \delta_{\xi}^+ \phi + u^- \delta_{\xi}^- \phi + v^+ \delta_{\eta}^+ \phi + v^- \delta_{\eta}^- \phi = Pe^{-1} (\delta_{\xi\xi} \phi + \delta_{\eta\eta} \phi) \quad (5.2.4)$$

where

$$u^+ = \frac{1}{2} (\tilde{u} + |\tilde{u}|) ; \quad u^- = \frac{1}{2} (\tilde{u} - |\tilde{u}|)$$

$$\delta_{\xi}^+ \phi_{ij} = \frac{\phi_{ij} - \phi_{i-1,j}}{\Delta \xi} ; \quad \delta_{\xi}^- \phi_{ij} = \frac{\phi_{i+1,j} - \phi_{ij}}{\Delta \xi}$$

$$\delta_{\xi\xi} \phi_{ij} = \frac{\phi_{i+1,j} - 2\phi_{ij} + \phi_{i-1,j}}{(\Delta \xi)^2}$$

with similar expressions for v^+ and δ_{η}^+ , etc.

These difference equations were solved using the Gauss-Seidel iterative method. The Schwarz alternating procedure (described in Section 3.6) was used for solving on overlapping grids.

A subroutine was written to solve (5.2.4) on a uniform rectangle. The solution on every grid was calculated by this routine. Only boundary conditions, step sizes $\Delta \xi$ and $\Delta \eta$, and the angle of rotation needed to be supplied. The numerical method and/or the differential equation can be modified by changing only the solver routine.

5.3 Adaptive Procedure

The adaptive process is similar to that used in the one-dimensional calculations described in Section 4.3. The primary differences are the additional procedures required to handle rotated, overlapping rectangles.

The nearest-neighbor algorithm is used to cluster the flagged points, and an ellipse-generated rectangle fit to each cluster. (These methods are described in Chapter 2.) If some part of a new rectangle falls outside the problem boundaries, its size is reduced while holding

the angle of rotation fixed. Unenclosed flagged points are then fit with a boundary-aligned rectangle.

The nominal step sizes on a refined grid are the parent mesh sizes divided by the refinement ratio R . The actual size is evaluated by adjusting the nominal step size such that there is an even number of mesh lengths in each coordinate direction. This is required to calculate 2h solutions. As a result, nominal mesh sizes are usually adjusted a small amount. The mesh size is adjusted, rather than the rectangle's size, since the latter may have already been adjusted to make the rectangle fall within the problem boundaries.

Processing of newly created grids to accommodate the overlapping grid-solution procedure is required. Grids are first marked to indicate whether they are isolated or overlap another grid. Overlapping grids are then sorted into disjoint, overlapping sets. The order in which grids are visited in the Schwarz solution procedure is then determined by recording relative positions of the grids in each disjoint set. The coordinates of grid-boundary segments that are internal to other grids are evaluated and stored. This information is used by the Schwarz driver to update boundary conditions and check for convergence. (The Schwarz solution process is described in the next section.)

Boundary conditions and initial guesses for fine grids are interpolated from coarse grids using bilinear interpolation (3.8.2). If a fine grid point lies on the external boundary, exact boundary conditions are applied. This is important, because interpolation of problem boundary values from a coarse grid could deteriorate the overall accuracy. Bilinear interpolation is also used to update overlapping-grid boundary conditions during the Schwarz solution process.

An active calculation was also performed by making a small modification to the passive program. The adaptive procedure is similar to the passive method described above, except that, after a new level of grids has been added, an active solution is calculated using the procedure described in Section 3.5.

Because the solution error is estimated in the passive program, it was used as a refinement criterion for the active method, rather than

the truncation error, as recommended in Section 3.6. This substitution is justified for two reasons. First, the active calculation is made only for a comparison against the passive calculation. Second, in this problem, the solution error is a good indicator of where the truncation error is large; the substitution will not have a large effect on the overall performance of the method. Additionally, the maximum allowed error in the active calculation was larger than that used in the passive one.

5.4 Numerical Results

Equation (5.2.1) was solved on the square domain $x \in (0,10)$; $y \in (0,10)$, with a constant uniform flow field, and step initial conditions:

$$u' = \cos \theta ; \quad v' = \sin \theta$$

$$V = \sqrt{(u')^2 + (v')^2} = 1 \quad (5.4.1)$$

$$\phi(0, y') = \begin{cases} 1 & ; \quad y' \geq y_0 \\ 0 & ; \quad y' < y_0 \end{cases}$$

where θ is the flow angle with respect to the x-axis. As indicated in Fig. 5.1, the scalar ϕ diffuses as it is convected downstream.

For large Peclet numbers, diffusion in the streamwise direction can be neglected. Equation (5.2.1) then becomes the heat equation in streamline coordinates and an analytical solution can be constructed. The exact solution ϕ_e is:

$$\phi_e(x', y') = 0.5 + 0.5 \operatorname{erf} \left(\frac{\sqrt{Pe} \{ (y' - y_0) u' - x' v' \}}{2 \sqrt{(y' - y_0) v' - x' u'}} \right) \quad (5.4.2)$$

where y_0 is the y-coordinate of the step discontinuity at the upstream boundary $x = 0$.

The exact solution (5.4.2) was used for Dirichlet boundary conditions on all problem boundaries except $(0,y)$, where the step was applied.

An adaptive calculation was made with reference length $L = 1$, $Pe = 1000$ and $y_0 = 4$, corresponding to an 11° flow angle with respect to the x-axis.

The convergence criterion for stopping Gauss-Seidel (G-S) iterations was:

$$\Delta\phi_{GS} = \max_{ij} |\phi_{ij}^n - \phi_{ij}^{n-1}| < 10^{-4} \quad (5.4.3)$$

For overlapping grids, Schwarz iterations were stopped when:

$$\Delta\phi_{SZ} = \max_k |\phi^m(\gamma_k) - \phi^{m-1}(\gamma_k)| < 10^{-4} \quad (5.4.4)$$

was satisfied for all grids k in the overlapping set, where γ_k are the internal grid boundaries. With Eqs. (5.4.3-4), all points in the flowfield satisfy the same criterion at convergence.

The passive adaptive calculation proceeded as follows. A solution was calculated on the initial 40×40 grid. The mesh size in both directions was doubled, an initial guess was interpolated from the base grid, and another solution was calculated. The pointwise solution error was estimated using (3.7.3a) with $p = 1$. Points having estimated solution error $> 10^{-3}$ were flagged. The initial region needing refinement is indicated in Fig. 5.2a.

The flagged points were then fit with the rectangles shown in Fig. 5.2b. The size of the rotated rectangle was reduced so that it was completely contained within the problem boundaries. The boundary-aligned rectangles were then fit to the remaining unenclosed flagged points. A refinement ratio of two was used. Boundary and initial conditions for these grids were interpolated from the solution on the base grid.

A solution was then generated on this set of three rectangles using the Schwarz method. We describe the solution procedure; refer to Fig. 5.2b for the notation. A converged solution was first calculated

on $G_{1,1}$. Boundary conditions along the internal boundary $\gamma_{2,1}$ were interpolated from the solution just calculated; then a solution was calculated on $G_{1,2}$. Boundary conditions along the internal boundary γ_3 were interpolated from this solution, and a solution was then calculated on $G_{1,3}$. This completed a forward sweep.

Boundary conditions along the boundary $\gamma_{2,3}$ were interpolated from the solution on $G_{1,3}$, and the solution then calculated on $G_{1,2}$. Finally, conditions along γ_1 are interpolated from $G_{1,2}$, and the solution calculated on $G_{1,1}$. This completed one Schwarz iteration. This procedure was repeated until the convergence criterion (5.4.4) was satisfied on all internal boundaries.

The mesh sizes on the three grids were then doubled and another solution calculated using the Schwarz procedure. (The alternating method converged in four Schwarz iterations on h grids, and two iterations for $2h$ grids.) The error was again estimated, and the resulting refinement region is indicated in Fig. 5.2c.

In all, three levels of refinement were used. The resulting grids are indicated in Fig. 5.2d. The size of the refined regions decreases as more refinement is added, and the central grid tends to align with the flow. Thus the method "homes in" on the shear layer. Note that the major source of error is the nonalignment of the upstream boundary grid with the flow. This error propagates downstream.

Uniform grid calculations were made with meshes finer than the initial coarse mesh used in the adaptive calculation. The rms solution errors were evaluated for both adaptive and uniform calculations:

$$e_{\text{rms}} = \frac{\sum_{i=2}^{i_{\text{max}}-1} \sum_{j=2}^{j_{\text{max}}-1} \left[(\phi_h)_{ij} - \phi_e(x'_i, y'_i) \right]^2}{(i_{\text{max}}-2)(j_{\text{max}}-2)} \quad (5.4.5)$$

where ϕ_e is the exact solution (5.4.2). The errors are plotted vs. cpu times in Fig. 5.3. (For adaptive calculations, the solution was interpolated onto the base grid, where the error was then evaluated.)

We do not describe the active calculation in any detail here; the procedure is similar to that described in Section 8.2.3 for the backstep

AD-A169 471

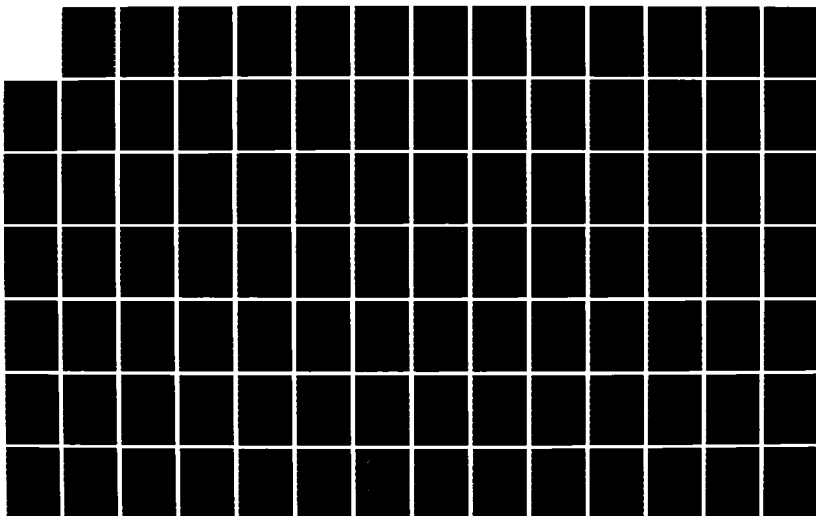
ADAPTIVE GRID TECHNIQUES FOR ELLIPTIC FLUID-FLOW
PROBLEMS(U) STANFORD UNIV CA CENTER FOR LARGE SCALE
SCIENTIFIC COMPUTATION S C CARUSO ET AL. DEC 85
CLASSIC-85-10 N00014-82-K-0335

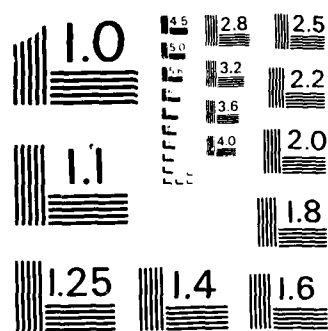
2/3

UNCLASSIFIED

F/G 20/4

NL





MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

problem, except for the refinement criterion. For this calculation, the maximum allowed error was 5×10^{-2} (compared to 10^{-3} for the passive calculation). The calculation was made with two levels of refinement, and the generated rectangles were smaller in size, compared to the rectangles at the same level in the passive calculation. This is a consequence of using a larger error criterion. The rms errors in the active solution were also calculated and are plotted in Fig. 5.3.

Figure 5.3 illustrates that both adaptive methods are considerably more efficient than uniform grid calculations. For fixed cost, the adaptive error is smaller or, alternatively, for fixed error, the adaptive calculation costs less. For example, for an accuracy of 3.5×10^{-2} (as indicated in the figure), the passive method runs approximately 15 times faster than the uniform-grid calculation. The trend of the passive curve indicates that its advantage increases as higher accuracy is required.

For a single level of refinement, the active calculation is found to be somewhat more efficient than the passive one--a result of the less conservative error criterion in the active method. However, there is a crossover in performance for two levels of refinement; the passive method becomes more efficient. The crossover occurs because the active method expends additional work updating the outer solutions, which do not need to be updated. For high levels of accuracy, the passive method is more efficient for this problem.

5.5 Conclusions

These calculations have shown that the passive method applied to the linear convection-diffusion equation is accurate and efficient when compared to uniform fine-grid calculations. The adaptive overhead (calculation of 2h solutions, error estimation, and grid generation) is included in the cost. The advantage of the adaptive method increases as higher accuracy is required.

Refined rectangles are automatically aligned with the flowfield, thereby minimizing numerical diffusion. Linear interpolation of boundary conditions was found to provide sufficient accuracy. The use of upwind differencing was important for providing accuracy outside the shear layer and also for reliable grid-refinement information.

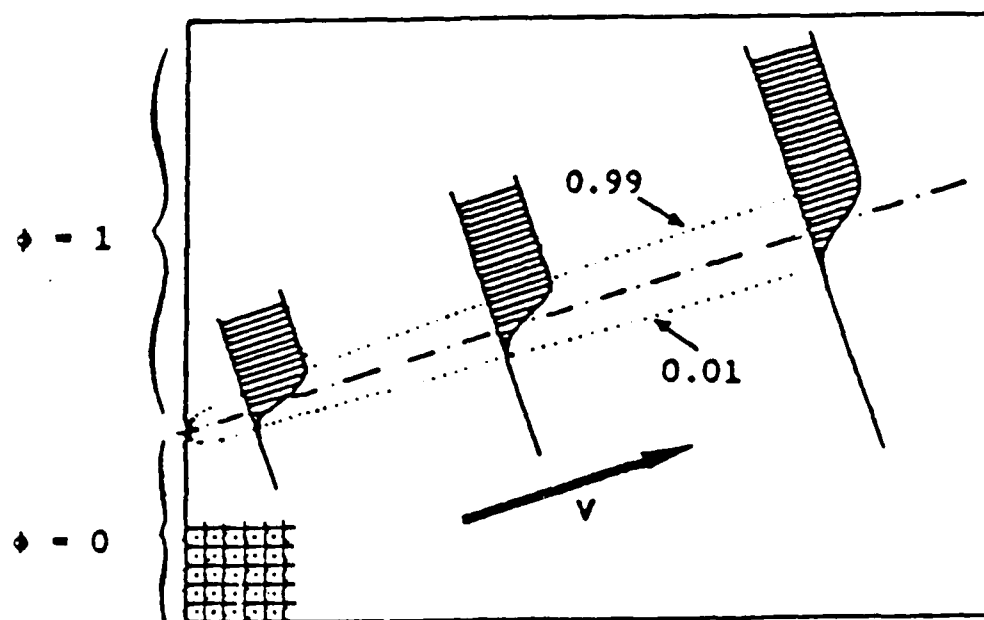


Fig. 5.1. Schematic of linear convection-diffusion problem.

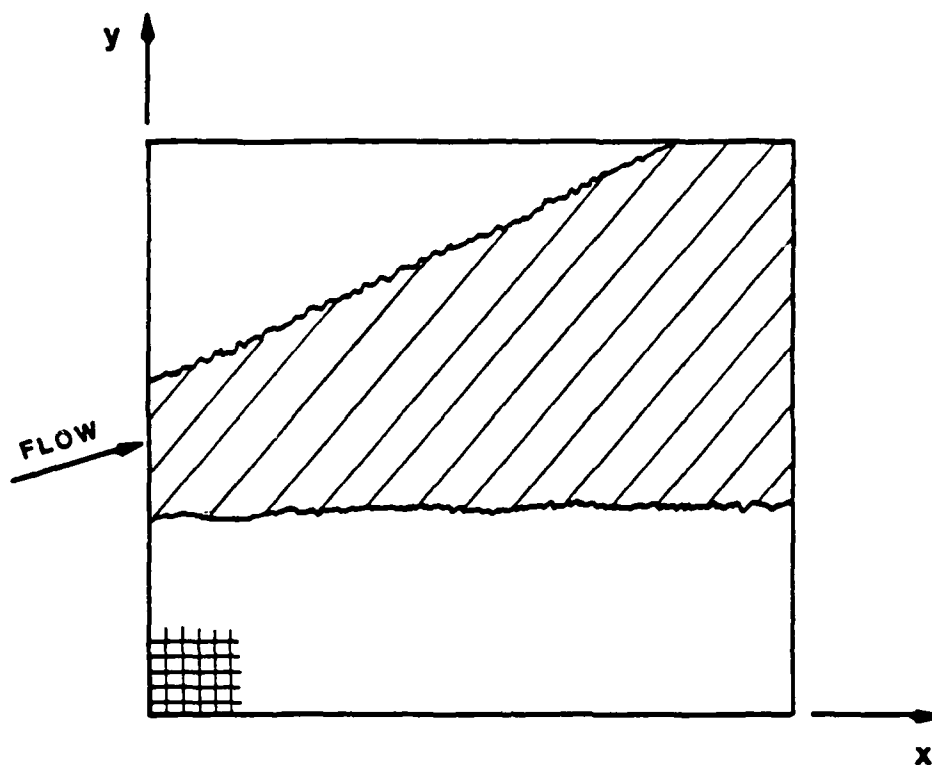


Fig. 5.2(a). Initial refinement region.

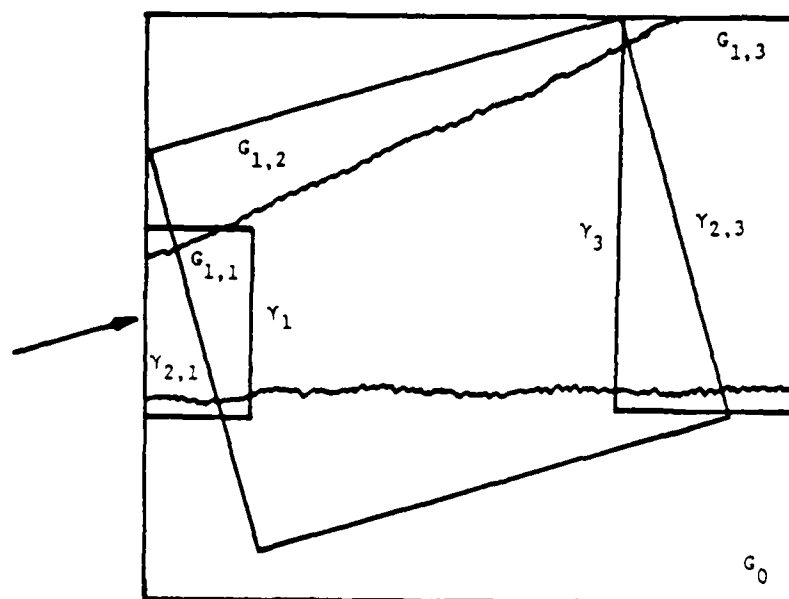


Fig. 5.2(b). First-level refined grids.

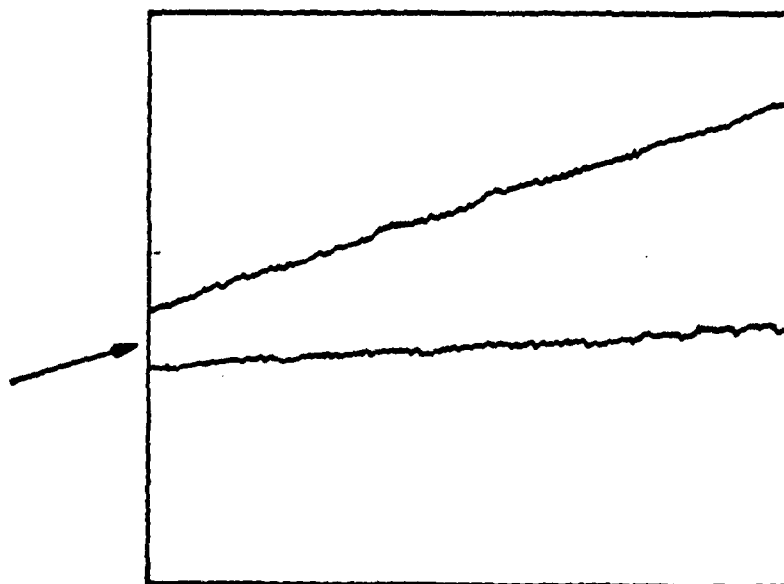
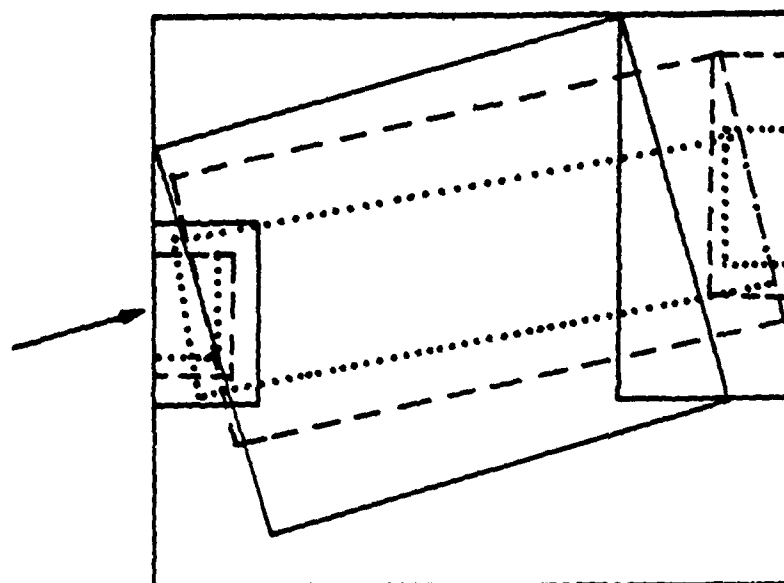


Fig. 5.2(c). First-level refinement region.



—— 1st level grids
 --- 2nd level grids
 3rd level grids

Fig. 5.2(d). Resulting refined grids.

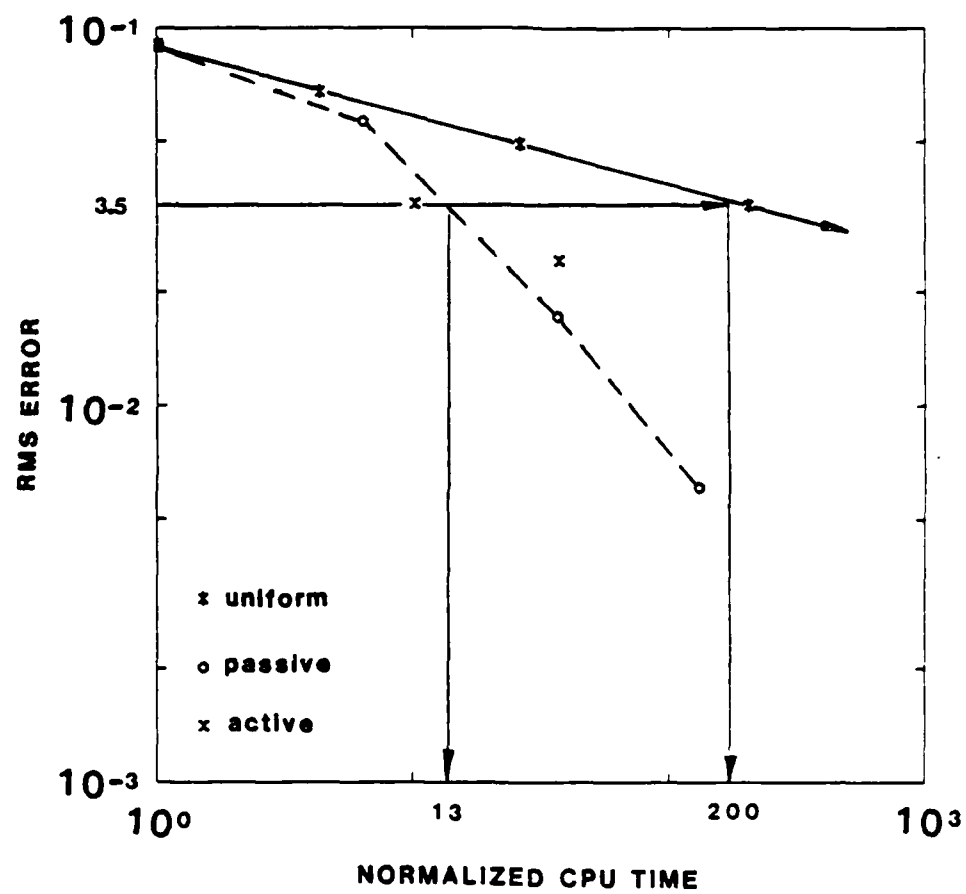


Fig. 5.3. Adaptive vs. uniform grid performance.

Chapter 6

NUMERICAL SOLUTION OF THE NAVIER-STOKES EQUATIONS

As mentioned previously, our main interest is flows described by the steady, laminar or Reynolds-averaged, incompressible Navier-Stokes equations. In this chapter we present the governing differential equations and the numerical methods used to solve these equations. These methods will be embedded in the adaptive method.

6.1 Navier-Stokes Equations

6.1.1 The Incompressible Navier-Stokes Equations

For completeness, we present the unsteady equations, which are incompletely parabolic. The steady equations are elliptic and are obtained by dropping the time derivatives.

In Cartesian tensor notation, the time-dependent, incompressible, Navier-Stokes (momentum) equations are:

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_j u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (6.1.1)$$

where $u_i(\underline{x}, t)$ is the local fluid velocity, ρ is the density, p is the pressure, and ν the viscosity. The continuity equation solved in conjunction with (6.1.1) is:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (6.1.2)$$

which requires the velocity field to be divergence-free.

The Navier-Stokes equations can be nondimensionalized in the following way. Using L and V as reference length and velocity scales, respectively, the dimensionless variables,

$$\tilde{x}_i = x_i / L$$

$$\tilde{u}_i = u_i / V$$

$$\tilde{p} = p/\rho V^2$$

$$\tau = t/(L/V)$$

are substituted into (6.1.1) and after rearranging obtain:

$$\frac{\partial \tilde{u}_i}{\partial \tau} + \frac{\partial \tilde{u}_j \tilde{u}_i}{\partial \tilde{x}_j} = - \frac{\partial \tilde{p}}{\partial \tilde{x}_i} + \text{Re}^{-1} \frac{\partial^2 \tilde{u}_i}{\partial \tilde{x}_j \partial \tilde{x}_j} \quad (6.1.3)$$

where Re is the Reynolds number, $\text{Re} = VL/\nu$.

Equation (6.1.3) governs both laminar and turbulent flows. However, at large enough Reynolds numbers (the exact value depending on the geometry), the flow becomes turbulent and is no longer steady.

Turbulent flows are difficult to compute because of their unsteadiness and because they possess a wide range of relevant physical scales. Accurate resolution of these scales requires using a prohibitively large number of grid points, even for moderate Reynolds numbers. Consequently, the direct solution of (6.1.1) for turbulent flows requires supercomputers and is restricted to relatively low Reynolds numbers. Such calculations are referred to as direct or full simulations (see, e.g., Ferziger, 1983).

6.1.2 Reynolds-Averaged Equations

Although turbulent flows are unsteady, time-averaged quantities (velocity, pressure, etc.) can be defined. To derive the averaged equations for these quantities, the velocity is decomposed into a mean and fluctuating part:

$$u_i = \bar{u}_i + u'_i \quad (6.1.4)$$

where \bar{u}_i and u'_i denote the mean and fluctuating velocities, respectively, and $\bar{u}'_i = 0$. The pressure is similarly decomposed. Substituting this decomposition into (6.1.1) and (6.1.2) and time-averaging results in the Reynolds-averaged Navier-Stokes equations yields

$$\frac{\partial \bar{u}_i}{\partial \tau} + \frac{\partial \bar{u}_j \bar{u}_i}{\partial \tilde{x}_j} = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial \tilde{x}_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial \tilde{x}_j \partial \tilde{x}_j} - \frac{\partial \overline{u'_i u'_j}}{\partial \tilde{x}_j} \quad (6.1.5)$$

and an averaged continuity equation

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0 \quad (6.1.6)$$

These equations describe the mean flow field \bar{u} and pressure \bar{p} , which are steady if $\partial \bar{u} / \partial t = 0$.

The term $-\rho \overline{u'_i u'_j}$ is called the Reynolds stress, and it represents the influence the turbulence exerts on the mean flow. Auxiliary equations must be supplied for the Reynolds stress in order to solve (6.1.5) and (6.1.6)

An eddy (or turbulent) viscosity relationship is a common constitutive equation for the Reynolds stress. It has the form:

$$\overline{u'_i u'_j} = -\nu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (6.1.7)$$

where a relation for ν_T , the turbulent viscosity, must be specified.

Equation (6.1.5) can be nondimensionalized in the same manner as Eq. (6.1.1). The resulting equation is similar to (6.1.3), except that the Reynolds number is replaced by an "effective" Reynolds number:

$$Re_{eff} = \frac{VL}{\nu_{eff}}$$

where $\nu_{eff} = \nu_T + \nu$.

If the eddy-viscosity model is used, the laminar and averaged equations have the same form and therefore are similar from a computational standpoint. The major difference is that ν_T varies spatially and is generally much larger than ν . Consequently, $Re_{eff}^{-1} \gg Re^{-1}$. This distinction is important, because the difficulty of computing these flows varies inversely with the effective Reynolds number.

6.1.3 Eddy-Viscosity Models

To complete our discussion of the turbulence equations, we briefly discuss some models for ν_T . A more detailed discussion of these models can be found in Rodi (1980) or the earlier review of Reynolds (1976).

The simplest expression for ν_T is the zero-equation model:

$$\nu_T = c\ell \left[\left(\frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_j}{\partial x_j} \right) \left(\frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \bar{u}_j}{\partial x_j} \right) \right]^{1/2} \quad (6.1.8)$$

where c is a constant and ℓ is a given length scale.

A more complex relation for ν_T is the one-equation model:

$$\nu_T = ck^{1/2}\ell \quad (6.1.9)$$

where $k = \frac{1}{2} \overline{u'_i u'_i}$ is the turbulent kinetic energy and is described by a transport equation similar to (6.1.11) below. c and ℓ have the same meaning as in (6.1.8).

In the popular two-equation k - ϵ model,

$$\nu_T = ck^2/\epsilon \quad (6.1.10)$$

where ϵ represents the dissipation of turbulent energy. c is a constant; k and ϵ are governed by transport equations of the form:

$$\frac{\partial k}{\partial t} + \frac{\partial \bar{u}_j k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + P - \epsilon \quad (6.1.11)$$

$$\frac{\partial \epsilon}{\partial t} + \frac{\partial \bar{u}_j \epsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\frac{\nu_T}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + c_{\epsilon_1} \frac{\epsilon}{k} P - c_{\epsilon_2} \frac{\epsilon^2}{k} \quad (6.1.12)$$

where σ_k , σ_ϵ , c_{ϵ_1} , c_{ϵ_2} are constants and P is the production

$$P = \nu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_j}$$

Note the similarity between the laminar equation (6.1.1), Reynolds-averaged equation (6.1.5), and the k and ϵ transport equations (6.1.11) and (6.1.12). All of them can be written in the following general form, for the steady case:

$$\frac{\partial \bar{u}_j \phi}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\Gamma_\phi \frac{\partial \phi}{\partial x_j} \right) + S_\phi \quad (6.1.13)$$

Equation (6.1.13) is elliptic, and the same numerical methods can be applied to all the equations. Our adaptive technique has been applied only to the steady, laminar equations. However, by modifying the flow solver, it can be extended to apply to the averaged turbulent equations, including any of the various eddy-viscosity models discussed above.

6.1.4 Steady, 2-D, Laminar Equations

Our adaptive procedure was applied to the 2-D, steady, laminar, incompressible equations. In Cartesian coordinates, the dimensionless x-momentum equation is:

$$\frac{\partial u^2}{\partial x} + \frac{\partial uv}{\partial y} = -\frac{\partial p}{\partial x} + \text{Re}^{-1} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (6.1.14a)$$

The y-momentum equation is:

$$\frac{\partial uv}{\partial x} + \frac{\partial v^2}{\partial y} = -\frac{\partial p}{\partial y} + \text{Re}^{-1} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (6.1.14b)$$

The continuity equation is:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (6.1.16)$$

This set is complete, but there is no explicit relationship for the pressure.

An equation for p can be derived by taking the divergence of (6.1.14) and using continuity to simplify. To derive this equation, first rewrite the momentum equations (6.1.14) as:

$$\frac{\partial p}{\partial x} = M^x \quad (6.1.16a)$$

$$\frac{\partial p}{\partial y} = M^y \quad (6.1.16b)$$

adding $\partial/\partial x$ (6.1.16a) to $\partial/\partial y$ (6.1.16b) results in a Poisson equation for the pressure:

$$\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2} = \frac{\partial M^x}{\partial x} + \frac{\partial M^y}{\partial y} = S_p \quad (6.1.17)$$

The source term, S_p , can be simplified using continuity, giving:

$$\nabla^2 p = - \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + 2 \left(\frac{\partial v}{\partial x} \right) \left(\frac{\partial u}{\partial y} \right) \right] \quad (6.1.18)$$

Equations (6.1.14) with (6.1.17) are an alternative description of incompressible flows.

Equations (6.1.14)-(6.1.15) are invariant under rotation, and consequently so is (6.1.17). These equations are therefore directly applicable to arbitrarily rotated grids.

6.2 Staggered Grid

The Navier-Stokes equations are solved on a uniform, rectangular, staggered grid (first proposed by Harlow and Welch, 1965). On this grid, each dependent variable is defined at a different set of locations, as indicated in Fig. 6.1. Pressure nodes are located at the cell centers, u , the x-component of velocity at the midpoints of the horizontal sides, and v , the y-component of velocity at the midpoints of the vertical sides.

This grid can be viewed as a composite of three grids, one for each variable. For example, u_{ij} , v_{ij} , and p_{ij} are all located at different points, as shown in Fig. 6.1.

For incompressible flows, the staggered grid has some important advantages over a nonstaggered grid (on which all dependent variables are defined at the same locations). As seen in Fig. 6.1b, the staggered grid has no pressure nodes located on its boundaries. Consequently, the pressure does not have to be explicitly specified there; this is not the case for a nonstaggered grid. This is an advantage, since boundary conditions for the pressure are not normally known. A Neumann boundary condition for pressure is implicitly contained in the numerical method, as discussed below.

The staggered grid is also more accurate than a nonstaggered grid, as the finite differences are taken over shorter distances. They are

therefore more accurate than centered differences on a nonstaggered grid (see Fig. 6.2).

Furthermore, on a nonstaggered grid, only every second pressure node is coupled (see Fig. 6.2b). As a result, an oscillating or checkerboard pressure solution is possible (see discussion on pp. 115-117, Patankar, 1980). This cannot occur on a staggered grid.

A disadvantage of the staggered grid is that the boundary cells are different from the normal cells, as indicated in Fig. 6.1b. Careful treatment is required to maintain accuracy at these points (see Section 6.3.2).

6.3 Finite Differences

In this section, we present the finite difference approximations used for the momentum equations. The treatment of the pressure equation is given in Section 6.4 below.

Standard methods for solving differential equations approximate derivatives by finite differences (FD) constructed from local Taylor expansions. An alternative technique is the finite volume (FV) method, which approximates the integrated conservation equations. We use the latter.

We demonstrate the application of the FV method to the x-momentum equation (6.1.14a); the y-momentum equation is handled in a similar manner.

Consider the finite volume centered around u_{ij} shown in Fig. 6.3. Equation (6.1.14a) is integrated over this volume; the volume integrals are converted to surface integrals using the divergence theorem, and the line integrals are evaluated using the mean-value theorem. The result is an exact integral-conservation equation given in terms of face-average values, designated e, w, n, and s (east, west, etc.):

$$\begin{aligned} & \left[u_e^2 - u_w^2 + p_e - p_w - Re^{-1} \left(\frac{\partial u}{\partial x} \Big|_e - \frac{\partial u}{\partial x} \Big|_w \right) \right] \cdot \Delta y \\ & + \left[u_n v_n - u_s v_s - Re^{-1} \left(\frac{\partial u}{\partial y} \Big|_n - \frac{\partial u}{\partial y} \Big|_s \right) \right] \cdot \Delta x = 0 \end{aligned} \quad (6.3.1)$$

Equation (6.3.1) represents the conservation of u-momentum in the finite volume (momentum theorem).

Next, the average face fluxes are approximated in terms of the neighboring grid-point values. The formula used to define the face values determines the resulting difference approximations; these are discussed next.

6.3.1 Pressure Difference

The pressure gradient is the simplest term to deal with, since the pressure grid points are located at the centers of the cell faces, as noted in Fig. 6.3. This results in the equivalent centered, second-order FD approximation:

$$\left. \frac{\partial p}{\partial x} \right|_{i+1/2,j} \approx \frac{\delta p}{\delta x} = \frac{p_{i+1,j} - p_{ij}}{\Delta x} \quad (6.3.2)$$

6.3.2 Diffusion Terms

For the diffusive fluxes, second-order central differences are also used, e.g.,

$$\left. \frac{\partial u}{\partial y} \right|_n \approx \frac{\delta u}{\delta y} \Big|_n = \frac{u_{ij+1} - u_{ij}}{\Delta y} \quad (6.3.3a)$$

$$\left. \frac{\partial u}{\partial y} \right|_s \approx \frac{\delta u}{\delta y} \Big|_s = \frac{u_{ij} - u_{i-1,j}}{\Delta y} \quad (6.3.3b)$$

Subtracting (6.3.3b) from (6.3.3a) and dividing the result by Δy gives the second-order central difference for the second derivative:

$$\left. \frac{\delta^2 u}{\delta y^2} \right|_{ij} = \frac{u_{ij+1} - 2u_{ij} + u_{i-1,j}}{(\Delta y)^2} \quad (6.3.3c)$$

As noted in the previous section, the staggered grid is nonuniform at horizontal boundaries for the u-grid; consequently, (6.3.3) has to be modified for boundary cells (refer to Fig. 6.4). A common approximation at the lower boundary is:

$$\left. \frac{\delta u}{\delta y} \right|_{i,1} = \frac{u_{i,1} - u_{i,0}}{\Delta y/2} \quad (6.3.4)$$

Subtracting (6.3.4) from (6.3.3a) and dividing by Δy gives the following approximation for $\left. \frac{\partial^2 u}{\partial y^2} \right|_{i,1}$:

$$\left. \frac{\partial^2 u}{\partial y^2} \right|_{i,1} = \frac{2u_{i,0} - 3u_{i,1} + u_{i,2}}{(\Delta y)^2} \quad (6.3.5)$$

However, the accuracy of this approximation is $O(1)$; i.e., it is not consistent with the differential equation. Since $\partial^2 u / \partial y^2$ is large near solid walls, it is important to approximate this term accurately.

A better approach is to use the first-order-accurate approximation:

$$\left. \frac{\partial^2 u}{\partial y^2} \right|_{i,1} = \frac{2u_{i,0} - 3u_{i,1} + u_{i,2}}{\frac{3}{4}(\Delta y)^2} \quad (6.3.6)$$

or the second-order approximation:

$$\left. \frac{\partial^2 u}{\partial y^2} \right|_{i,1} = \frac{16u_{i,0} - 25u_{i,1} + 10u_{i,2} - u_{i,3}}{5(\Delta y)^2} \quad (6.3.7)$$

To test these approximations, a fully developed channel-flow problem was calculated. This problem has the exact solution:

$$\begin{aligned} u &= y(1-y) \\ v &= 0 \end{aligned} \quad (6.3.8)$$

$$0 \leq x \leq 1 ; \quad 0 \leq y \leq 1$$

Using either (6.3.6) or (6.3.7), the converged numerical solution agrees with the exact solution. Using the approximation (6.3.5), significant solution error results near the grid boundaries. We use the first-order scheme (6.3.6) in most of our calculations.

6.3.3 Convective Terms

First, we briefly review the problems associated with approximating the convective terms. Our discussion is directed toward steady flows, although similar difficulties are encountered in time-dependent problems.

Central differencing (CD) of the convective term has two associated difficulties. First, at large Reynolds numbers, CD may produce solutions that have large spatial oscillations or "wiggles". An illustration of this in one dimension was given in Fig. 4.2e.

It can be shown (see, e.g., pp. 24-26, Peyret and Taylor, 1983) that, if CD is used to solve the one-dimensional, linear, convection-diffusion equation,

$$\bar{u} \frac{\partial \phi}{\partial x} = \nu \frac{\partial^2 \phi}{\partial x^2} ; \quad \bar{u}, \nu \text{ const.} \quad (6.3.8)$$

oscillations in the solution will occur unless

$$Re_{\Delta x} = \frac{|\bar{u}| \Delta x}{\nu} < 2 \quad (6.3.9)$$

$Re_{\Delta x}$ is called the cell Reynolds number. This restriction may require using prohibitively fine meshes for large Reynolds numbers.

Even though (6.3.9) is strictly valid only for (6.3.8), this restriction is often applied to the momentum equations, which are nonlinear, multidimensional, and have source terms. However, one cannot prove that (6.3.9) must be satisfied for CD to give smooth solutions to the momentum equations.

We have found smooth solutions to the laminar back-step problem using CD for cell Reynolds numbers as high as 150 (see Chapter 8). Kim and Moin (1985) have found similar results. The apparent difficulty is that streamwise diffusion is not important in this flow. For further discussion, see Ferziger (1986).

The second difficulty with CD is that instability may occur when iterative methods such as Gauss-Seidel are used to solve the system of difference equations. If the difference equations are written in matrix form:

$$Au = b \quad (6.3.10)$$

then a sufficient condition for convergence of the Gauss-Seidel method is that A be diagonally dominant, i.e.,

$$|a_{ii}| \geq \sum_{\substack{j=1 \\ j \neq i}}^{j_{\max}} |a_{ij}| ; \quad i = 1, i_{\max} \quad (6.3.11)$$

where a_{ij} are the elements of A . (This criterion also holds for many other iterative methods.) CD applied to (6.3.8) or the momentum equations gives diagonally dominant matrices only if $Re_{\Delta x} < 2$.

The possibility of obtaining spatial oscillations and the instability of iterative methods have led researchers to search for alternative differencing approximations. Upwind differencing (UD) was the first remedy tried. UD always gives diagonally dominant matrices and smooth solutions; however, it introduces excessive numerical diffusion, as discussed below. The search for better methods remains an active research area.

We discuss the schemes which we use next.

Central Differencing

Second-order central differencing for the convective terms is equivalent to assuming that u is piecewise linear between the four neighboring points of u_{ij} . On a uniform grid, this means that face fluxes are averages of the adjacent grid-point values. For example, the formulas for the horizontal fluxes in (6.3.1) are:

$$u_e^2 = \left(\frac{u_{ij} + u_{i+1,j}}{2} \right)^2 \quad (6.3.12a)$$

$$u_w^2 = \left(\frac{u_{ij} + u_{i-1,j}}{2} \right)^2 \quad (6.3.12b)$$

For the vertical fluxes, v must be averaged, i.e.,

$$u_n v_n = \left(\frac{u_{ij} + u_{ij+1}}{2} \right) \left(\frac{v_{ij} + v_{i+1,j}}{2} \right) \quad (6.3.13)$$

Upwind Differencing

In upwind differencing, the fluxes at the cell faces are taken from the "upwind" grid point values. The "wind direction" is determined by the sign of the velocity at the cell center.

For the horizontal fluxes, the sign of u determines the wind direction, e.g.,

$$\left. \begin{aligned} u_e^2 &= u_{1j}^2 \\ u_w^2 &= u_{i-1,j}^2 \end{aligned} \right\} u_{1j} \geq 0$$

(6.3.14)

$$\left. \begin{aligned} u_e^2 &= u_{i+1,j}^2 \\ u_w^2 &= u_{1j}^2 \end{aligned} \right\} u_{1j} < 0$$

The sign of v at the cell center determines the wind direction for the vertical fluxes. This value, \bar{v} , is found by averaging, e.g.,

$$\bar{v} = \frac{1}{4} (v_{1j} + v_{i+1,j} + v_{i,j-1} + v_{i+1,j-1}) \quad (6.3.15)$$

The vertical fluxes are then obtained from

$$u_n = \begin{cases} u_{1j} & ; \quad \bar{v} \geq 0 \\ u_{i+1,j} & ; \quad \bar{v} < 0 \end{cases} \quad (6.3.16a)$$

$$v_n = \frac{v_{1j} + v_{i+1,j}}{2} \quad (6.3.16b)$$

and similarly for u_s and v_s .

UD is first-order accurate, but, more importantly, the leading term of the truncation error is proportional to the second derivative. For positive u_{1j} , an upwind approximation to $u(\partial u / \partial x)$ on a regular (non-staggered) grid has the following truncation error:

$$u \frac{\partial u}{\partial x} \Big|_{ij} = u_{ij} \frac{u_{ij} - u_{i-1,j}}{\Delta x} + \frac{u_{ij} \Delta x}{2} \frac{\partial^2 u}{\partial x^2} \Big|_{ij} + O(\Delta x^2) \quad (6.3.17)$$

The truncation error thus introduces numerical diffusion. At even moderate Reynolds numbers, the numerical diffusion can greatly exceed the physical diffusion. Consequently, if the grid is not sufficiently refined, the result may be an inaccurate, overly diffused solution.

Hybrid Differencing

The next two methods are composites of central and upwind differencing. The first scheme, attributed to Spalding (1972), uses central differencing at a grid point if the magnitude of the local cell Reynolds number, $Re_{\Delta x} < 2$, and upwind differencing otherwise. (For two-dimensional problems, $Re_{\Delta y} = \frac{|v| \Delta y}{\nu}$ is used as the criterion for the y-derivatives.) Additionally, where $Re_{\Delta x} > 2$, the (physical) diffusion term in (6.3.1) is discarded to partially offset the numerical diffusion introduced by the upwind differencing.

This scheme has the stability and monotonicity of the pure upwind method. However, in most practical computations, $Re_{\Delta x} \gg 2$ in most of the flow. Consequently, this method is only marginally better than pure UD.

The second hybrid scheme is Patankar's power-law method (Patankar, 1980, and 1981). It is based on the exact solution to (6.3.8) and is similar to the exponential scheme of Allen and Southwell (1955).

Face values are found in the following manner. Equation (6.3.8) is integrated along the line $x_L < x < x_R$, with the boundary conditions

$$\begin{aligned} \phi(x_L) &= \phi_L \\ \phi(x_R) &= \phi_R \end{aligned} \quad (6.3.18)$$

to give:

$$\frac{\phi - \phi_L}{\phi_R - \phi_L} = \frac{\exp(\overline{Re}(x-x_L)/\Delta x) - 1}{\exp(\overline{Re}) - 1} \quad (6.3.19)$$

where

$$\overline{Re} = \frac{\overline{u} \Delta x}{\nu}$$

$$\Delta x = x_R - x_L$$

Equation (6.3.19) is used to evaluate the velocities at the cell faces. For example, using

$$\phi(x_i) = u_{i,j}$$

$$\phi(x_{i+1}) = u_{i+1,j}$$

the face velocity u_e is found by evaluating (6.3.19) at $x = x_{i+1/2}$, with \overline{u} taken as the average of $u_{i,j}$ and $u_{i+1,j}$ at the previous time step or iteration.

Patankar uses a power-law fit to the exponential (6.3.19) that is cheaper to evaluate. The velocities for all cell faces are evaluated in the same manner and substituted in the integral equation (6.3.1). Patankar's method is stable for all cell Reynolds numbers. It is appropriate for approximating (6.3.8); however, its accuracy when applied to the momentum equations is not clear.

The power-law scheme is similar to Spalding's hybrid method, but it has a gradual transition from central to upwind differencing in the vicinity of $Re_{\Delta x} = 2$.

QUICK Differencing

The last method discussed is the upwind-biased, QUICK difference scheme (Leonard, 1979a,b). (QUICK is an acronym for Quadratic Upstream Interpolation for Convective Kinematics.) Face fluxes are determined by quadratically interpolating the solution from the two adjacent, upwind grid points and the one adjacent downwind point at each face.

For the horizontal fluxes, the resulting approximation is (Shyy, 1985):

$$u_e^2 - u_w^2 = \begin{cases} \frac{3}{8} (u_{i+1,j}^2 + u_{ij}^2) - \frac{7}{8} u_{i-1,j}^2 + \frac{1}{8} u_{i-2,j}^2 & ; \quad u_{ij} \geq 0 \\ -\frac{1}{8} u_{i+2,j}^2 + \frac{7}{8} u_{i+1,j}^2 - \frac{3}{8} (u_{ij}^2 + u_{i-1,j}^2) & ; \quad u_{ij} < 0 \end{cases} \quad (6.3.20)$$

The vertical fluxes are found in a similar manner.

Dividing (6.3.20) by Δx gives a second-order-accurate, noncentered FD for the convective term $\partial u^2 / \partial x$. The overall method remains second-order accurate as second-order approximations are also applied to the diffusion terms.

QUICK's accuracy is comparable to central differencing and more accurate than the upwind or hybrid schemes (Leschziner, 1980, Leschziner and Rodi, 1981, Durst and Pereira, 1983). However, the method does not always give diagonally dominant matrices, and convergence is not guaranteed (Han et al., 1981). QUICK can also give oscillatory solutions, although they are generally smoother than those obtained with CD. Our experience confirms these observations (see Chapter 8).

6.4 SIMPLER Solution Technique

Patankar's SIMPLER (Semi-Implicit Method for Pressure-Linked Equations Revised) method is used to solve the systems of finite difference approximations to the momentum and Poisson equations. The method is described in Patankar (1980); we give a summary and an alternative interpretation of the scheme in this section.

The procedure is iterative. Beginning with an initial velocity field, which does not necessarily satisfy continuity, we do the following:

1. Calculate the pressure by approximately solving a Poisson equation (6.4.2).
2. Approximately solve the linearized momentum equations (6.4.3) for velocity, using the pressure from Step 1.
3. Calculate the pressure correction by approximately solving a Poisson equation (6.4.7). (The pressure correction is formulated to make the updated velocity in Step 4 satisfy continuity.)

4. Calculate the velocity correction (6.4.6) and add it to the velocity from Step 2. (The velocity correction is proportional to the gradient of the pressure correction.)
5. Check for convergence:

$$\max_{i,j} |\underline{u}_{ij}^{n+1} - \underline{u}_{ij}^n| < \epsilon$$

Steps 1-5 are cyclically repeated. As the solution converges, the divergence of the velocity field, the pressure correction, and the velocity correction are all driven towards zero. The final velocity and pressure fields satisfy the discretized momentum and Poisson equations, respectively, to within a convergence criterion.

The equations solved in Steps 1-3 are linear systems and are approximately solved at each step using the same iterative solution method. We describe this method in Section 6.4.4 below.

The SIMPLER technique was heuristically formulated. However, Patankar does not clearly indicate what forms of the differential equations are being approximated (e.g., the type of linearization for the momentum equations, the specific form of the Poisson equation, etc.). to clarify this, we next present an alternative interpretation of the method.

6.4.1 Pressure Calculation

In Step 1, the pressure is evaluated from an approximation to the Poisson equation (6.1.17). The source term is evaluated by first calculating the right-hand sides of (6.1.16a,b) using the current velocities u^n and v^n . For example,

$$M_{ij}^x = - \left[\frac{\delta u^2}{\delta x} + \frac{\delta uv}{\delta y} - \text{Re}^{-1} \left(\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} \right) \right]_{ij}^n \quad (6.4.1)$$

M^x is evaluated at all u-grid points, M^y at all v-grid points (see Fig. 6.5).

The divergence of $M = [M^x, M^y]^T$ and the Laplacian of p are both evaluated using second-order central differences, resulting in a Poisson equation for the pressure:

$$\frac{\delta^2 p^{n+1}}{\delta x^2} + \frac{\delta^2 p^{n+1}}{\delta y^2} = \left[\frac{\delta M^x}{\delta y} + \frac{\delta M^y}{\delta y} \right]^n \quad (6.4.2)$$

where

$$\left. \frac{\delta^2 p}{\delta x^2} \right|_{ij} = \frac{p_{i+1,j} - 2p_{ij} + p_{i-1,j}}{(\Delta x)^2}$$

$$\left. \frac{\delta M^x}{\delta x} \right|_{ij} = \frac{M_{ij}^x - M_{i-1,j}^x}{\Delta x}$$

with similar expressions for $\delta^2 p / \delta y^2$ and $\delta M^y / \delta y$. The staggered grid is more accurate than a nonstaggered one, because it results in centered differencing over one mesh length.

At convergence, the pressure satisfies the Neumann condition (6.1.16a) on vertical boundaries and (6.1.16b) on horizontal boundaries.

6.4.2 Momentum Calculation

The discrete momentum equations solved in Step 2 are linearized in the following manner:

$$\frac{\delta u^n u^*}{\delta x} + \frac{\delta u^* v^n}{\delta y} = - \frac{\delta p^{n+1}}{\delta x} + \text{Re}^{-1} \left(\frac{\delta^2 u^*}{\delta x^2} + \frac{\delta^2 u^*}{\delta y^2} \right) \quad (6.4.3a)$$

$$\frac{\delta u^n v^*}{\delta x} + \frac{\delta v^n v^*}{\delta y} = - \frac{\delta p^{n+1}}{\delta y} + \text{Re}^{-1} \left(\frac{\delta^2 v^*}{\delta x^2} + \frac{\delta^2 v^*}{\delta y^2} \right) \quad (6.4.3b)$$

where u^* and v^* are the unknowns and the difference formulas are the same as in (6.4.1). These can also be written in operator form:

$$L_x u^* = - \frac{\delta p^{n+1}}{\delta x} \quad (6.4.4a)$$

$$L_y v^* = - \frac{\delta p^{n+1}}{\delta y} \quad (6.4.4b)$$

Underrelaxation of the velocities is required to ensure convergence of the overall scheme. u^* and v^* are simultaneously underrelaxed during

the approximate solution of (6.4.3); the procedure is described in Section 6.4.4 below.

6.4.3 Velocity and Pressure Corrections

In Step 4, the velocity corrections u' and v' are added to the velocities calculated in (6.4.3) to give the updated velocities:

$$\begin{aligned} u^{n+1} &= u^* + u' \\ v^{n+1} &= v^* + v' \end{aligned} \quad (6.4.5)$$

The corrections are formulated to make the updated velocity field satisfy continuity.

The velocity corrections are related to the pressure correction p' by eliminating all of the contributions from the velocities at the neighboring points in the momentum equations (6.4.3) to give:

$$\begin{aligned} u' &= A^u \frac{\delta p'}{\delta x} \\ v' &= A^v \frac{\delta p'}{\delta y} \end{aligned} \quad (6.4.6)$$

where A^u and A^v depend on the difference scheme.

A similar correction is made in Chorin's projection method (Chorin, 1968). There, the pressure is used instead of p' , and the velocity correction is interpreted as an operator that projects an arbitrary velocity field, u^* , onto a divergence-free field, u^{n+1} . p' plays a similar role in SIMPLER.

The equation for p' is found by requiring that u^{n+1} satisfy continuity. Inserting (6.4.5) with (6.4.6) into a discretized version of continuity (6.1.2) results in the following Poisson equation for p :

$$\frac{\delta^2 p'}{\delta x^2} + \frac{\delta^2 p'}{\delta y^2} = - \left(\frac{\delta u^* / A^u}{\delta x} + \frac{\delta v^* / A^v}{\delta y} \right) \quad (6.4.7)$$

The same differencing is used for (6.4.7) as for (6.4.2).

If the velocity is prescribed on the boundaries, the boundary condition for (6.4.7) is $p' = 0$. Intermediate values of u^{n+1} will

not satisfy continuity, because (6.4.7) is only approximately solved. At convergence, $u^{n+1} = u^* = u^n$, continuity is satisfied and p' vanishes everywhere.

6.4.4 Solution of the Linear Systems

An iterative method is used to solve the linear systems (6.4.2-3) and (6.4.7) in Steps 1-3. The scheme is essentially an ADI method; it is called a "line-by-line" method by some authors. One iteration consists of making line Gauss-Seidel relaxations in one coordinate direction, followed by similar relaxations in the other direction.

For example, the solution at points along a vertical line (indicated by the dots in Fig. 6.6) are simultaneously evaluated using the current neighboring solution values (at locations indicated by the x's). The equations for the dotted variables are tridiagonal and are solved via the Thomas algorithm. Lines are solved in succession in one direction, then in the other direction.

Since the SIMPLER scheme is iterative, it is not necessary to solve the linear systems exactly at each step. We perform one line-by-line iteration of each system for each SIMPLER iteration. Sweeps are made in the flow direction to get the best convergence rate.

As noted above, u^* and v^* are underrelaxed simultaneously when (6.4.3a,b) are solved using the line-by-line method. Next, we describe how the underrelaxation is performed.

The difference equation (one row of (6.4.4a)) for the velocity u_p at some grid point P can be written:

$$a_p u_p^* = \sum a_{nb} u_{nb}^* + f \quad (6.4.8)$$

where the subscript nb denotes the neighboring grid points of P ; the summation is to be taken over all neighboring points.

Equation (6.4.8) can be rewritten as:

$$u_p^* = u_p^n + \left[\frac{\sum a_{nb} u_{nb}^* + f}{a_p} - u_p^n \right] \quad (6.4.9)$$

where u_p^n is the velocity from the previous SIMPLER iteration. The term in the brackets can be regarded as the change in u_p^* for the current iteration.

To reduce this change, an underrelaxation factor, ω , where $0 \leq \omega \leq 1$ is introduced:

$$u_p^* = u_p^n + \omega \left[\frac{\sum a_{nb} u_{nb}^* + f}{a_p} - u_p^n \right] \quad (6.4.10)$$

Equation (6.4.10) can be rewritten:

$$\frac{a_p}{\omega} u_p^* = \sum a_{nb} u_{nb}^* + f + \frac{(\omega-1)}{\omega} a_p u_p^n \quad (6.4.11)$$

The relaxation is introduced by scaling the diagonal terms a_p and adding the last term in (6.4.11) to the right-hand side of (6.4.4) before performing the line-by-line iteration.

The relaxation factor must be determined experimentally; we have used $\omega = 0.85$, as recommended by Zebib (1984) for most calculations. However, at higher Re and smaller grid sizes, ω must be reduced to keep the SIMPLER method from diverging.

6.5 Implementation of Central Differencing for $Re_{\Delta x} > 2$

As discussed in Section 6.3.3, central differencing (CD) can destabilize the solution method when $Re_{\Delta x} > 2$. The matrix equations representing the x- and y-momentum equations are not diagonally dominant when $Re_{\Delta x}$ and $Re_{\Delta y} > 2$, respectively. Without diagonal dominance, the line-by-line method becomes unstable, and the solution blows up.

A stable method can be constructed using the defect-correction method (see. e.g., Stetter, 1978, Hemker, 1981, Auzinger and Stetter, 1982), which we describe below. Note that this method does not eliminate oscillations in the solution. Oscillations are artifacts of the difference equations, not the solution method.

Assume that we wish to solve the differential equation:

$$Lu = f$$

Denote L_h^1 as a difference method that gives diagonally dominant matrices, and L_h^2 as a more accurate method, which does not necessarily give diagonally dominant matrices. A solution can be computed that satisfies L_h^2 by means of the following iterative procedure:

$$L_h^1 u^0 = f \quad (6.5.1a)$$

$$L_h^1 u^{n+1} = f + (L_h^1 u^n - L_h^2 u^n) \quad (6.5.1b)$$

An iterative method can be used to approximately solve the linear system (6.5.1b) at any intermediate step, e.g., one line-by-line sweep. However, at convergence, the solution satisfies:

$$L_h^2 u^\infty = f \quad (6.5.2)$$

We apply the correction procedure (6.5.1) to the x- and y-momentum equations (6.4.4a,b) in the following manner:

$$L_x^1 u^* = -\frac{\delta p^{n+1}}{\delta x} + (L_x^1 u^n - L_x^2 u^n) \quad (6.5.3a)$$

$$L_y^1 v^* = -\frac{\delta p^{n+1}}{\delta y} + (L_y^1 v^n - L_y^2 v^n) \quad (6.5.3b)$$

where L_x^1 and L_y^1 represent Patankar's hybrid scheme and L_x^2 and L_y^2 are central differencing. The Poisson equation for pressure (6.4.2) is unchanged. The solution method is stable for all cell Reynolds numbers, and the converged solution is second-order accurate.

Any other difference method can be easily substituted into an existing program. One need only insert the correction $(L_h^1 u^n - L_h^2 u^n)$ on the right-hand side of the equation as a source term.

6.6 Numerical Conservation

The notion that numerical approximations to the fluid-flow equations should satisfy integral conservation equations to within round-off errors is an important and controversial one. In this section, we briefly review the concept and how it applies to single grid computations. We discuss the implications for local adaptive refinement in the next chapter.

We begin by defining numerical conservation. A scheme is said to be conservative if it satisfies a discrete version of the Gauss divergence theorem. For example, consider the continuity equation in vector form:

$$\nabla \cdot \underline{u} = 0 \quad (6.6.1)$$

Integrating (6.6.1) over a volume V , and making use of the divergence theorem allows us to write:

$$\int_V \nabla \cdot \underline{u} dV = \int_S \underline{u} \cdot \underline{n} ds \quad (6.6.2)$$

where S is the surface of the volume and \underline{n} the outward unit-normal to the surface.

Assume that we have the following difference scheme for (6.6.1):

$$\nabla_h \cdot \underline{u} = 0 \quad (6.6.3)$$

A discrete analog of the divergence theorem in 2-D is:

$$\sum_i \sum_j w_{ij}^1 (\nabla_h \cdot \underline{u})_{ij} \Delta x \Delta y = \sum_i \sum_j w_{ij}^2 u_n \Delta S \quad (6.6.4)$$

(interior points) (boundary points)

where u_n is the velocity normal to the boundaries, and w^1 and w^2 are weighting functions dependent on the numerical integration formulas. The scheme (6.6.3) conserves mass if (6.6.4) is satisfied exactly to within round-off error.

The quadrature method should be an inverse of the difference method (analogous to the integral and differential operators). The weighting functions w^1 and w^2 are usually taken to be unity, which corresponds to the trapezoidal rule or the mid-point rule, depending on whether the

integrated quantity is located at cell faces or cell centers, respectively. For $w^1 = w^2 = 1$, the difference equations (6.6.3) are summed over all grid points.

If the scheme conserves mass, all interior grid-point velocity differences cancel, and the remaining boundary terms can be rearranged to represent the surface integral in (6.6.4). Since velocity differences cancel, there are no numerical sources or sinks of mass in the interior of the domain.

A similar analysis can be made for the momentum equations. If all discrete volume integrals can be reduced to surface integrals, the scheme conserves momentum.

The construction of conservative schemes is simplified with the finite volume method, if the conservation form (6.1.1) of the differential equations is used as a starting point. (In the nonconservation form, the convective terms are written as $u_j(\partial u_i / \partial x_j)$.) If face-flux expressions are formulated such that the flux across a common face of adjacent control volumes is the same for both control volumes, then they will cancel when summed over interior grid points.

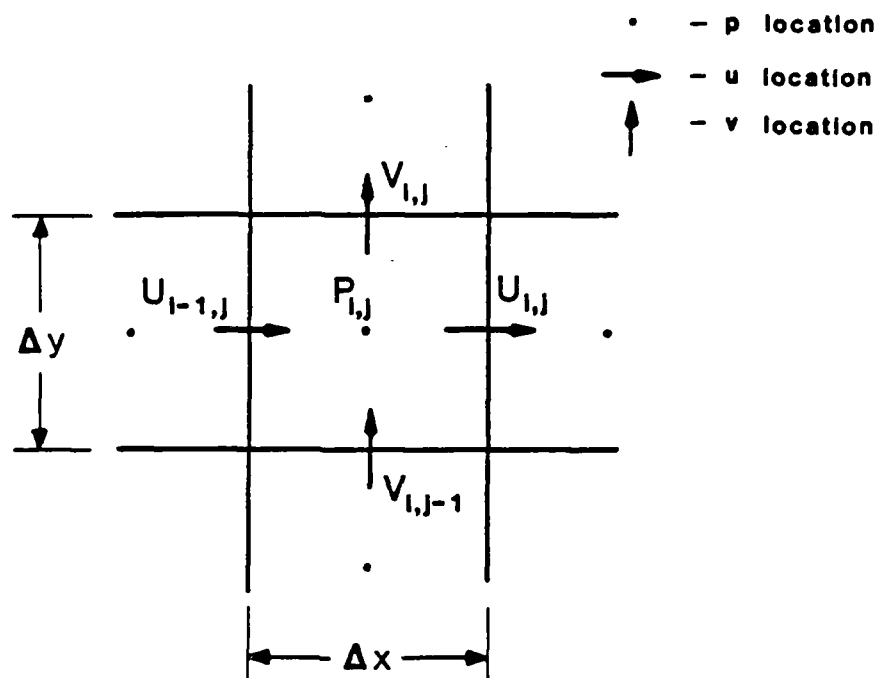
The integral fluid-flow equations are conservation statements for the various physical quantities: mass, momentum, energy, etc. The discrete integrals are not, in general, equal to their respective analytical counterparts. However, the quantities may be exactly conserved in a numerical scheme. Conservation and accuracy are separate issues. An accurate, nonconservative method will give the exact solution (including conservation) in the limit of vanishing mesh size, and consequently be conservative in the limit if the differential equation is conservative.

There has been considerable debate over whether methods need to be conservative or not. For shock calculations, it is well known that a conservative method can be used to compute the correct shock speed and strength. Instability has been attributed to a lack of conservation in some time-dependent schemes for viscous flows. However, there are also instances where a nonconservative method gave more accurate results than a conservative one. (Roache, 1982, pp. 32-33, reviews the early literature on this subject.) The consensus is that conservative methods should be used whenever possible.

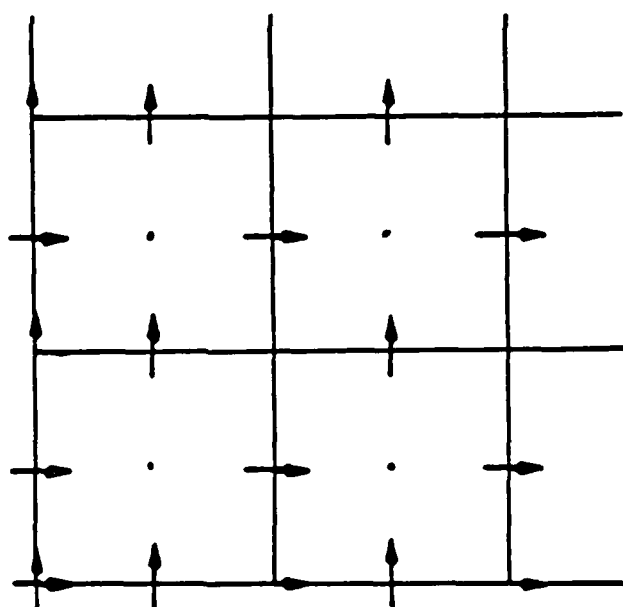
On a staggered grid, central differencing for all terms in the momentum equations conserves mass and momentum--and kinetic energy in the limit of infinite Reynolds numbers (Zabusky and Deem, 1971). But the scheme is conservative only if approximations similar to (6.3.4) are used for the diffusion term for boundary cells. The more accurate schemes (6.3.6) and (6.3.7) which we use make the method nonconservative.

We make one final point. Given a conservative method, it is necessary that the surface integral in (6.6.4) vanish for steady flows, if global conservation is to be maintained. For rectangular grids, this means that the normal velocities at all boundary grid points must sum to zero. We therefore took care to specify boundary conditions for the base-grid computations such that global continuity is satisfied.

In this chapter, we have described the solution techniques used for the steady, laminar equations. These methods are incorporated in our general flow solver, which is applicable to an arbitrarily rotated, uniform rectangle. This solver is used to calculate h and $2h$ solutions for the base grid and any refined grid in an adaptive computation.



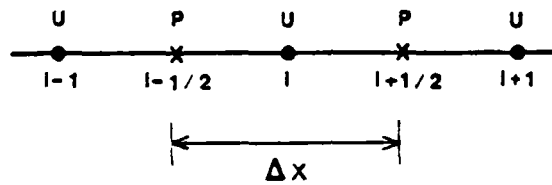
(a) Internal cell



(b) boundary cells

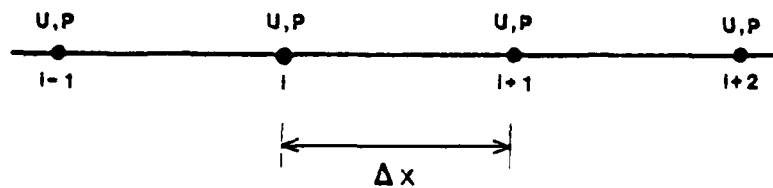
Fig. 6.1. Staggered grid geometry.

$$\left(\frac{dP}{dx}\right)_i \approx \frac{P_{i-1/2} - P_{i+1/2}}{\Delta x}$$



(a) staggered grid

$$\left(\frac{dP}{dx}\right)_i \approx \frac{P_{i+1} - P_{i-1}}{2\Delta x} \quad \left(\frac{dP}{dx}\right)_{i+1} \approx \frac{P_{i+2} - P_i}{2\Delta x}$$



(b) nonstaggered grid

Fig. 6.2. Centered differences on staggered and nonstaggered grids.

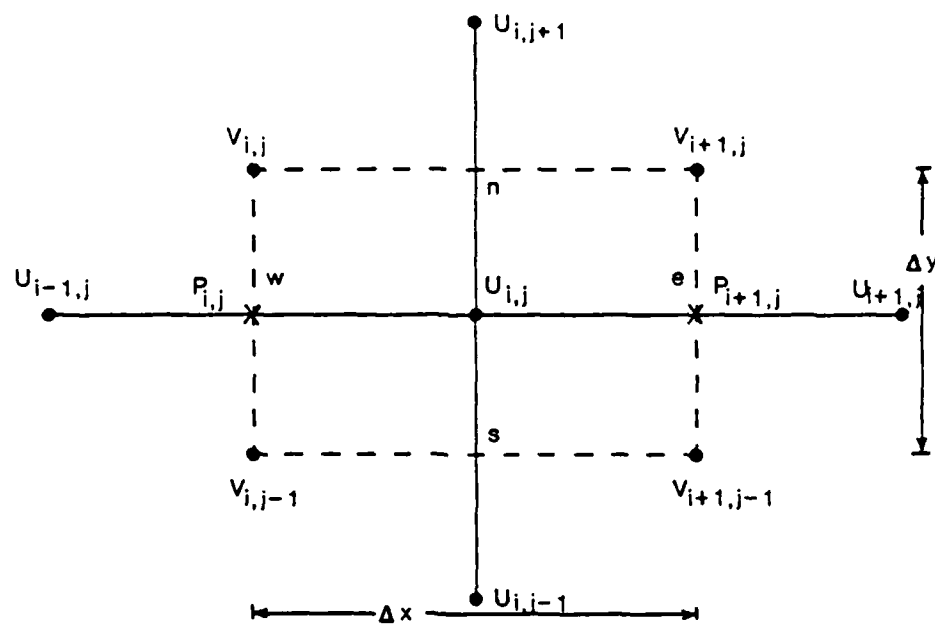


Fig. 6.3. Finite volume for u .

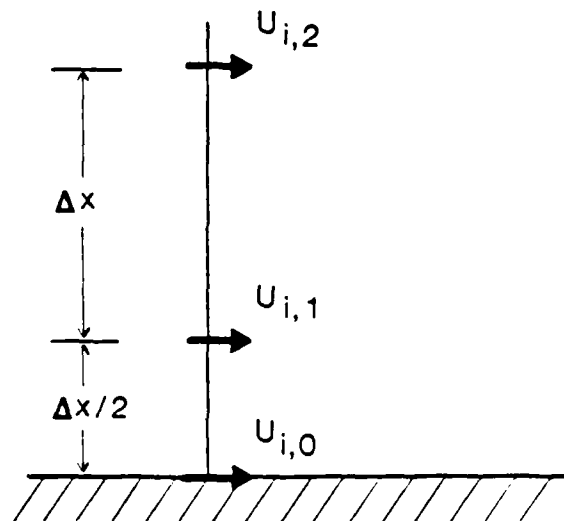


Fig. 6.4. Difference stencil for boundary cells.

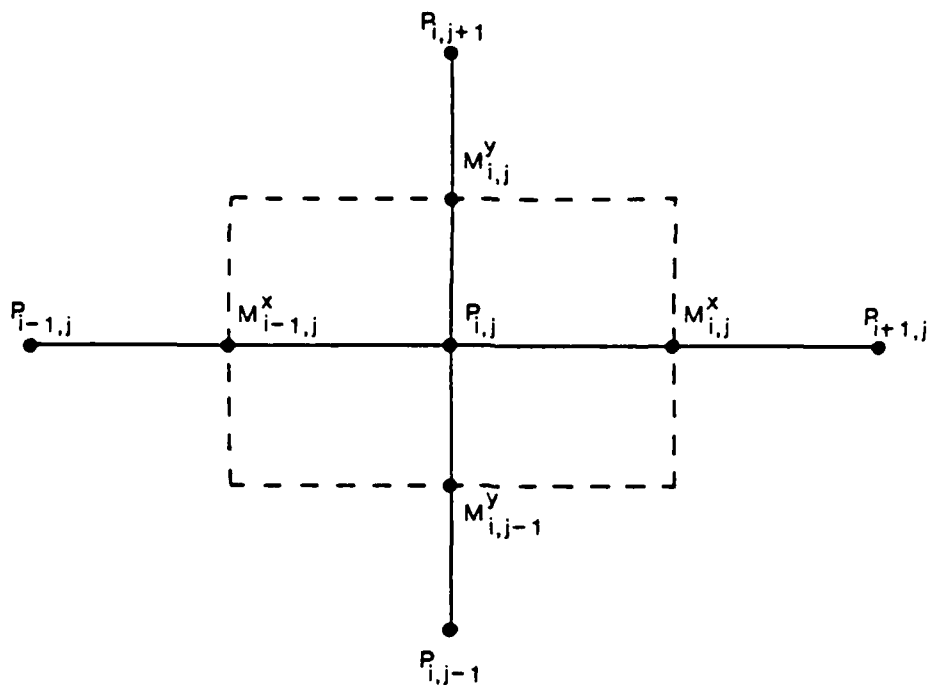


Fig. 6.5. Finite volume for p .

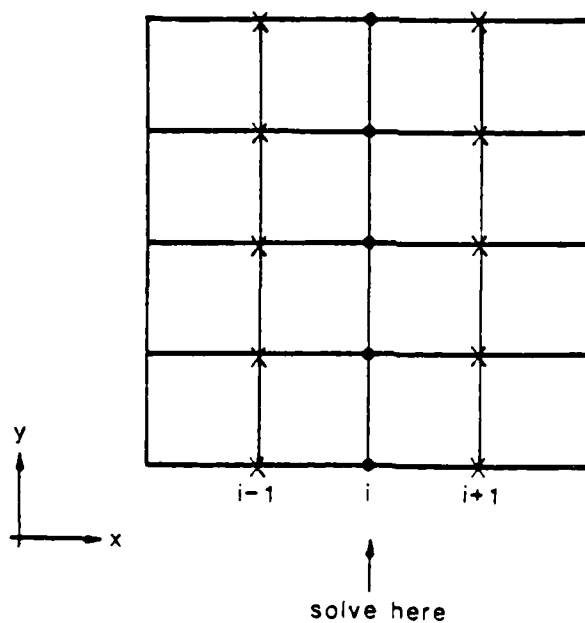


Fig. 6.6. Line-by-line solution method.

Chapter 7

ADAPTIVE NAVIER-STOKES SOLVER

In this chapter we describe the procedure for adaptively solving the steady, laminar, Navier-Stokes equations. We begin by summarizing the adaptive process, then follow with details of the active solution method, boundary conditions and conservation, and, finally, error-estimation method.

7.1 Summary of Adaptive Process

The adaptive method will be applied to the laminar backward-facing-step problem in Chapter 8. Because it is a strongly coupled problem, the active method is required. All refined grids are specified a priori to be boundary-aligned; rotated grids are not used. Finally, the refined grid boundaries are restricted to be colinear with parent grid lines, as indicated in Fig. 7.1.

Although our method is not restricted to nonrotated refinement, two factors influenced this choice. The primary factor is a consequence of the backstep geometry; use of rotated rectangles would decrease the adaptive efficiency. This is justified in Section 8.2.3. A secondary factor is the difficulty of conservatively interpolating boundary conditions for rotated grids; this is further discussed in Section 7.3.

With the exception of the solution method (active) and nonrotated refinement, the adaptive procedure is similar to that used for the linear convection-diffusion problem in Chapter 5. We summarize the adaptive solution process next, then discuss implementation details in the sections that follow.

The base-grid solution is calculated first. All velocity boundary conditions are Dirichlet, except at the outflow boundaries, where zero derivative conditions are applied. A solution on a doubled mesh is calculated with the same boundary conditions and used to estimate the truncation error via the procedure discussed in Section 3.7. Points having estimated truncation errors larger than a specified value are clustered; cluster(s) are fit with boundary-aligned, refined rectangle(s).

Boundary conditions for refined grids are all Dirichlet. Exact values are used wherever grid points lie on the problem boundary. For fine grid boundaries interior to the problem domain, values are interpolated from the parent grid using a procedure that conserves mass across the boundary (see Section 7.2 below). Initial guesses are bilinear interpolations from the parent grid.

An active solution is then calculated on the two-level grid system according to the algorithm outlined in Section 3.5. A solution is first calculated on the fine grid(s). Correction terms for the coarse grid are then evaluated and a solution recalculated on the coarse grid. Iteration between coarse and fine grids is repeated until internal fine-grid boundary values no longer change.

Because the method is iterative, the solution on each grid level is not required to be fully converged at intermediate steps. (The same approach is used with the momentum and Poisson equations in the SIMPLER method—see Section 6.4.4.) Rather, the solution may be iterated a fixed number of times (similar to multigrid methods) or to a given level of partial convergence, before switching grids. The iteration strategy is discussed in Chapter 8.

After the two-level solution has converged, its error is estimated by doubling the mesh sizes for all grids, and solving. The truncation error estimate is computed over the whole domain. Since the region of large truncation error decreases in size after each adaptation, the new level of refined grids is normally contained within those of the previous level.

7.2 Implementation of the Active Solution Method

We show how the active solution method is implemented for the laminar equations by applying it to the two-grid system shown in Fig. 7.2. Extension to more levels is straightforward.

We use the notation of Section 3.5. At convergence, the solution on the coarse grid G_0 in region Ω_1 should agree with the fine-grid solution. The solution consists of the velocity components u and v and the pressure p . However, as p can be determined from u and v ,

we require only that u and v agree on G_0 and G_1 on Ω_1 . As discussed in Section 3.5, $u_H = u_h$ on Ω_1 is enforced by calculating correction terms for the coarse grid.

If m indicates the active outer iteration number and n the SIMPLER (inner) iteration number, the correction terms are computed by modifying the right-hand sides of the coarse-grid momentum equations (6.5.3) in the following manner:

$$L_{x,H}^1 u_H^{n+1} = (1-\alpha_x) f_x^n + \alpha_x L_{x,H}^1 u_H^m \quad (7.2.1a)$$

$$L_{y,H}^1 v_H^{n+1} = (1-\alpha_y) f_y^n + \alpha_y L_{y,H}^1 v_H^m \quad (7.2.1b)$$

where

$$f_x^n = -\frac{\delta p^{n+1}}{\delta x} + (L_{x,H}^1 u_H^n - L_{x,H}^2 u_H^n)$$

$$f_y^n = -\frac{\delta p^{n+1}}{\delta y} + (L_{y,H}^1 v_H^n - L_{y,H}^2 v_H^n)$$

$$\alpha_x(x,y), \alpha_y(x,y) = \begin{cases} 1 & \text{if } (x,y) \in \Omega_1 \\ 0 & \text{if } (x,y) \in \Omega_0 - \Omega_1 \end{cases}$$

Note that the defect corrections are contained in f_x and f_y . (Recall that they are used to implement central differencing for the convective terms, as discussed in Section 6.5.) The parent grid-masking arrays α_x and α_y are initialized to zero and are modified when an offspring (refined grid) is created. On the finest grids, $\alpha_x = \alpha_y = 0$ everywhere.

The correction terms in (7.2.1) are evaluated by interpolating fine-grid solution values and using them in the coarse-grid operator L_H^1 according to the rules given in Section 3.5. Although u_h and v_h satisfy the more accurate L_h^2 operator on the fine grid, L_H^1 must be used for the corrections in order to give at convergence:

$$u_H = u_h \quad \text{where} \quad \alpha_x = 1$$

$$v_H = v_h \quad \text{where} \quad \alpha_y = 1$$

This can be verified by multiplying (7.2.1a) with $\alpha_x = 1$ by the inverse of L_H^1 and doing similarly for (7.2.1b).

7.3 Treatment of Boundary Conditions

Boundary conditions for refined grids are interpolated from coarse-grid solutions using a procedure that conserves mass. Additionally, when new grids are created, certain coarse-grid boundary conditions may require modification to maintain global continuity. In this section, we describe boundary conditions and discuss conservation and interpolation issues related to rotated grids.

7.3.1 Interpolation of Fine-Grid Boundary Conditions

As previously discussed, bilinear interpolation is normally used to transfer solution values between coarse and fine grids. However, interpolation can destroy numerical conservation at internal fine-grid boundaries.

Consider the west face of the central coarse-grid control volume in Fig. 7.1. The location of coarse-grid velocities are indicated by solid-head arrows, fine-grid velocities by open-head arrows. The fine grid is denoted by the dashed lines; the refinement ratio is 2. Since the fluid density is constant, the mass flux through the west face is proportional to the integral of the velocity over the face.

To conserve mass, the coarse-grid and fine-grid mass fluxes across the west face should be exactly equal. If the mid-point rule is used for integration, the coarse-grid mass flux is equal to the coarse-grid normal velocity at the face. The fine-grid mass flux is the sum of the two fine-grid velocities on the face.

If fine-grid velocities are linearly interpolated from the coarse grid, mass will not generally be conserved. However, linearly interpolating the coarse-grid velocity from the fine grid exactly conserves mass. Thus, linear interpolation from fine to coarse grids conserves mass, while coarse to fine interpolation does not. To be consistent,

the coarse to fine interpolation should be the inverse of the fine to coarse grid procedure.

Such a procedure was suggested by Berger (1984a), and is similar to methods used in TVD schemes for hyperbolic conservation laws (see, e.g., Van Leer, 1979). In this method, the normal velocity is assumed to vary linearly on the coarse-grid cell boundary, as indicated in Fig. 7.3. The slope is taken as the slope of the line connecting the two adjacent coarse-grid point values, as indicated in Fig. 7.3. Fine-grid normal velocities are determined by this line.

In other words, fine-grid normal velocities, $\bar{u}_h(x)$, are obtained from:

$$\bar{u}_h(x) = \left[\frac{u_H(x_j+H) - u_H(x_j-H)}{2H} \right] (x - x_j) + u_H(x_j) \quad (7.3.1)$$

for

$$x_j - h \leq x \leq x_j + h$$

Tangential velocities on internal boundaries are bilinearly interpolated from coarse grids; this does not affect the mass balance on the control volume.

An equation similar to (7.3.1) is used to interpolate v on horizontal faces (e.g., north and south faces in Fig. 7.1); u is bilinearly interpolated on these faces.

This scheme conserves only mass at the internal grid boundaries. It would be difficult to construct a method that also conserves additional quantities, e.g., momentum, kinetic energy, etc.; the necessity for using such a method is not clear. As shown in the next chapter, the scheme we use provides good accuracy and gives no stability problems when applied to the backstep flow.

7.3.2 Modification of Coarse-Grid Boundary Conditions

As mentioned previously, exact boundary conditions are applied if a fine-grid boundary lies on the problem boundary. There are also coarse-grid points on these "coincident" boundaries. If exact boundary values

are applied on all grid points on a coincident boundary, the numerical integral of the normal velocity on the boundary will be different on each grid; i.e., mass will not be conserved. To be conservative, all integrated mass fluxes should be equivalent.

Coincident boundaries are handled in the following manner. Exact values are applied on the finest grid, giving the most accurate approximation to the integrated mass flux. To maintain conservation, boundary values for coarse-grid points on a coincident boundary are linearly interpolated from the next finest grid.

In practice, when a new level of refined grids is created, boundary conditions are specified. At coincident boundaries, values for the next coarsest grid are obtained by linear interpolation from the fine grid; i.e., they are changed from their previous values. This is repeated for still coarser grids.

By using the conservative, coarse-to-fine-grid interpolation scheme (described in the previous section) and modifying coarse-grid boundary conditions at coincident boundaries, mass is conserved on the multilevel grid system.

7.3.3 Interpolation and Conservation for Rotated Grids

In this section, we discuss conservation and interpolation for rotated grids; further discussion can be found in Berger (1984b). From Fig. 7.4a, it is clear that conservative interpolation is complicated by the rotation. While an interpolation scheme that conserves mass is possible, the construction of a scheme that conserves additional quantities is more difficult. We shall therefore consider mass-conserving schemes.

Note that the schemes used with "patched" or zonal grids cannot be used for rotated grids. Adjacent patched grids do not overlap; rather, their boundaries interface along a common line. The interpolation is one-dimensional and therefore simpler compared to two-dimensional interpolation required for rotated, overlapping grids. See Hessenius and Rai (1984) or Rai (1985) for a discussion of interpolation methods used for patched grids.

Several interpolation approaches for rotated grids are possible; we outline a straightforward one. Consider interpolation of fine-grid boundary values along the line $a-a$ shown in Fig. 7.4b. The mass flux normal to the line $a-a$, \dot{m}_{a-a} , can be obtained from a mass balance on the triangular region, using the coarse-grid velocities u_H and v_H .

Assuming velocities are piecewise constant on cell faces gives:

$$\dot{m}_{a-a} = w_H(g+2h) = (fu_H + Hv_H) \quad (7.3.2)$$

where $(g+2h)$, f and H are the lengths of the triangle's sides.

\dot{m}_{a-a} is then to be distributed to the fine boundary grid cells #1, 2, 3 lying along line $a-a$, i.e.,

$$\dot{m}_{a-a} = \dot{m}_1 + \dot{m}_2 + \dot{m}_3 \quad (7.3.3)$$

If the flux is distributed according to individual cell areas, then

$$\dot{m}_1 = \dot{m}_2 = \frac{h}{(g+2h)} \cdot \dot{m}_{a-a} = hu_h^1 = hu_h^2 \quad (7.3.4)$$

and

$$\dot{m}_3 = \frac{g}{(g+2h)} \cdot \dot{m}_{a-a} \quad (7.3.5)$$

The fine-grid velocities u_h^1 and u_h^2 are obtained from (7.3.4). \dot{m}_3 is added to the flux obtained for the balance of cell #3, calculated using a similar procedure.

Coarse-to-fine grid interpolation could be performed in a similar manner. However, the calculation and distribution of mass fluxes depends on the assumed velocity variation on cell faces and on the quadrature method used to integrate the mass along the boundaries. (The midpoint rule is implicit in the procedure outlined above.) It may be possible to use arbitrary variations (e.g., constant, linear, quadratic, etc.) and integration methods, although this is not clear.

Fuchs (1985) uses a simpler approach. He solves the steady, laminar equations on two overlapping rectangular grids (similar to those shown in Fig. 3.6) with a multigrid method. For each grid, internal

boundary values are linearly interpolated from the other. Interpolated values are then "corrected" such that global continuity is satisfied on the respective grid. Fuchs does not indicate how the values are corrected; presumably they are either scaled or a small correction is added to each such that global continuity is satisfied. Fuchs notes that the corrections are second-order in magnitude. The computed results are only qualitatively evaluated (streamlines are plotted); however, the author concludes that the interpolation scheme works adequately.

As noted in Section 6.6, whether a method should be strictly conservative or not is a debated issue. We made calculations using non-conservative, bilinear interpolation for internal fine-grid boundary conditions and other calculations on single grids in which global continuity was not satisfied. The nonconservative calculations converged slower (on the order of 10-15%) and also were somewhat less accurate compared to similar conservative computations, but no stability problems were encountered with the nonconservative schemes.

7.4 Error Estimation

In this section, we describe the error-estimation method used for the steady, laminar equations.

Assume that an active solution has been calculated on a multilevel grid system. Let u_h and v_h represent this solution on all grid levels (h should be regarded as the finest mesh size). The meshes are doubled on all grids, and an initial guess for each is obtained by interpolating from its respective h -grid. An active solution u_{2h} and v_{2h} is then calculated.

For each grid, the truncation error estimate is calculated only at points not contained in finer-level grids. For example, for the grids shown in Fig. 7.2, the estimate is calculated at all points on G_1 and only at those points not in the region Ω_1 on G_0 .

The solution error estimate is calculated first. For example, the error in u_h is found by computing:

$$\tilde{e}_h^u(x_i, y_j) = \frac{u_h(x_i, y_j) - u_{2h}(x_i, y_j)}{2^p - 1} \quad (7.4.1)$$

at points on the h -grid. Since no h - and $2h$ -grid points are coincident (see Fig. 7.1), u_{2h} is linearly interpolated in order to evaluate the numerator in (7.4.1). With central differencing for all terms in the momentum equations, $p = 2$. With hybrid differencing for the convective terms, $p = 1$.

If a $3h$ - instead of a $2h$ -grid were used to compute the error estimates, interpolation would not be necessary, since every third point on the h -grid would be coincident with a $3h$ -grid point. However, this has the disadvantage that the error would be known at only $1/3$ of the grid points. Additionally, the $3h$ -estimate would be less accurate than a $2h$ -estimate. Results given in the next chapter show that our method provides good estimates of both solution and truncation errors.

The truncation error estimate $\tilde{\tau}^u$, as described in Section 3.7, is then evaluated by performing the explicit calculation:

$$\tilde{\tau}^u(x_i, y_j) = \left(L_{x,h}^p \left[u_h(x_i, y_j) + \tilde{e}_h^u(x_i, y_j) \right] + \frac{\delta p}{\delta x} \Big|_{ij} \right) \cdot (\Delta x \Delta y)_k \quad (7.4.2)$$

where $L_{x,h}^p$ and $\delta p / \delta x$ are the difference operators for the x -momentum equations in (6.4.5) used in calculating u_h . The estimate is scaled by Δx and Δy , the mesh sizes on the grid G_k , since we solve the integrated (finite volume) equations (6.3.1). (The scaling is merely a matter of programming convenience.) The truncation error for v_h is similarly evaluated.

This completes our description of the active adaptive method applied to the steady, laminar equations. We show the results of adaptive calculations of the laminar backstep in the next chapter.

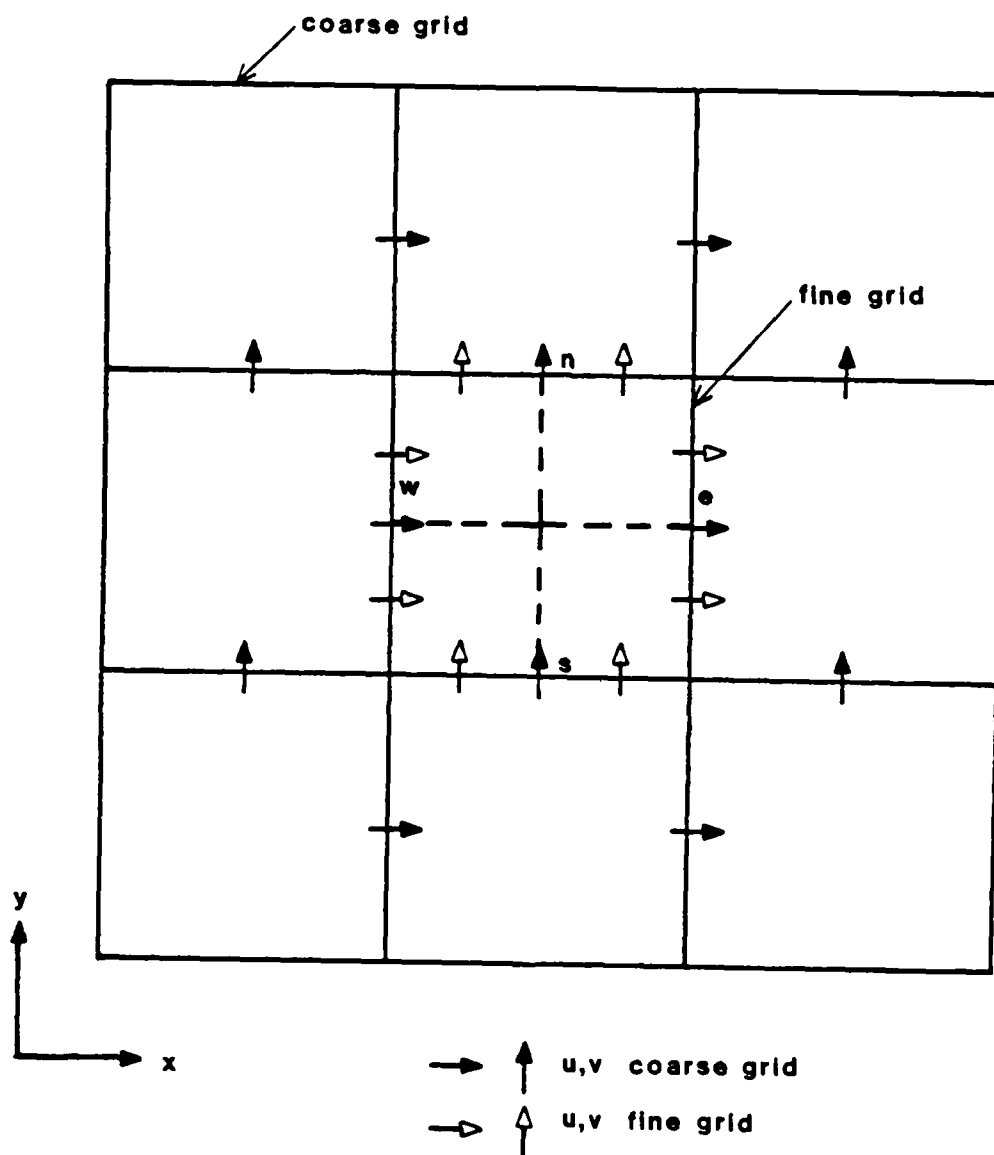


Fig. 7.1. Boundary-aligned refined-grid geometry.

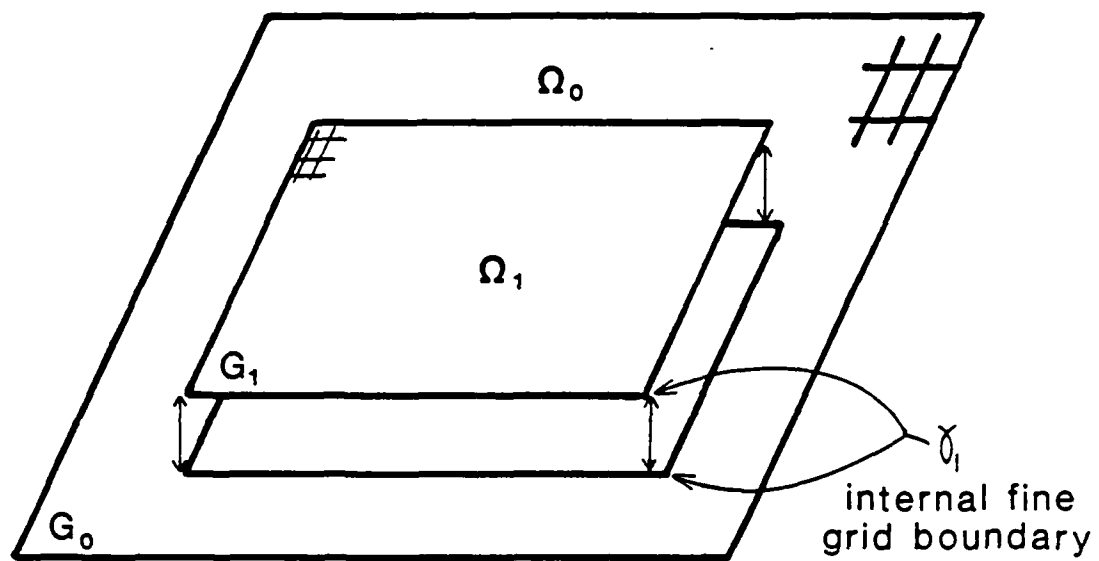


Fig. 7.2. Notation for active solution on two-level, 2-D grid system.

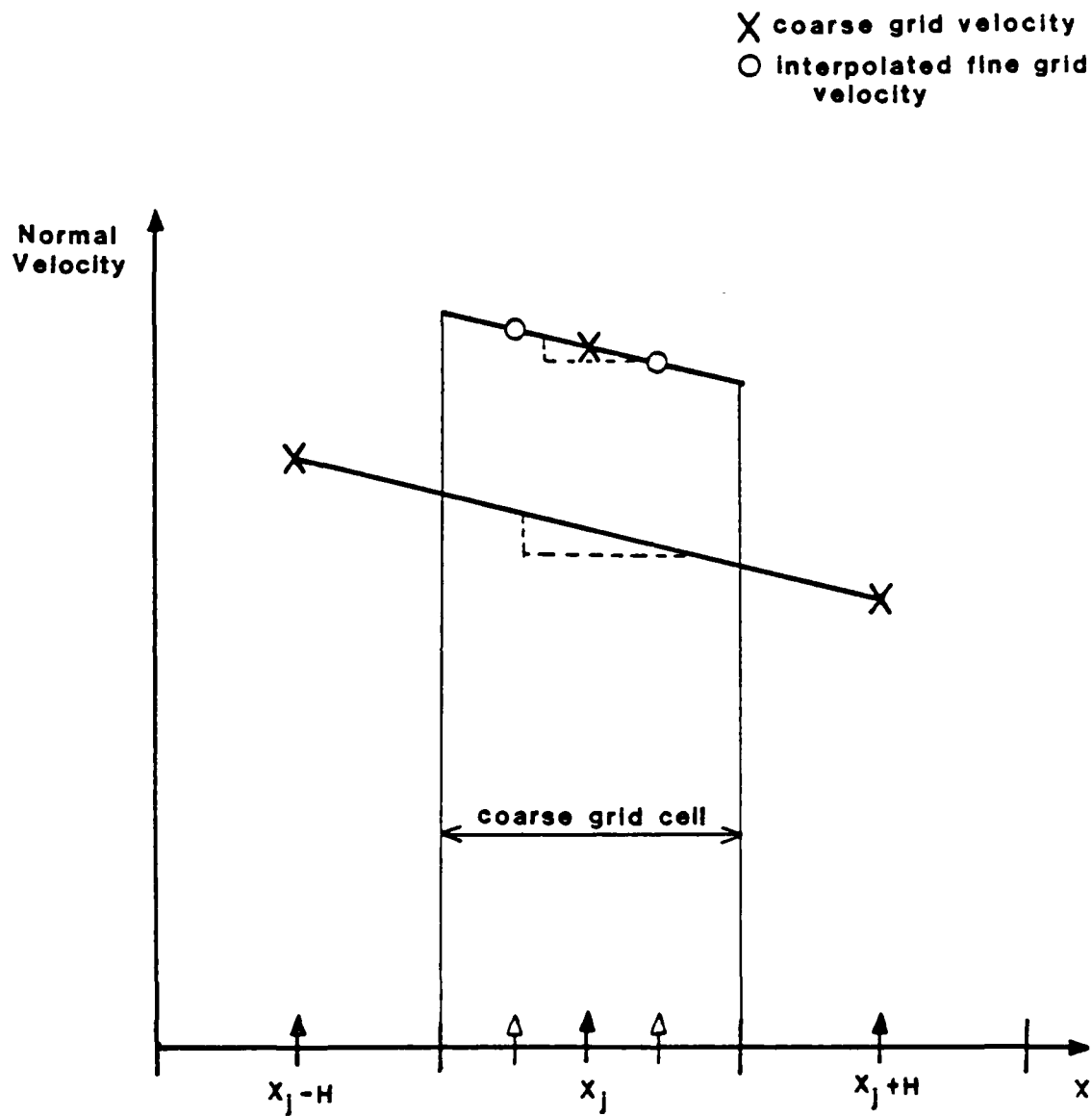
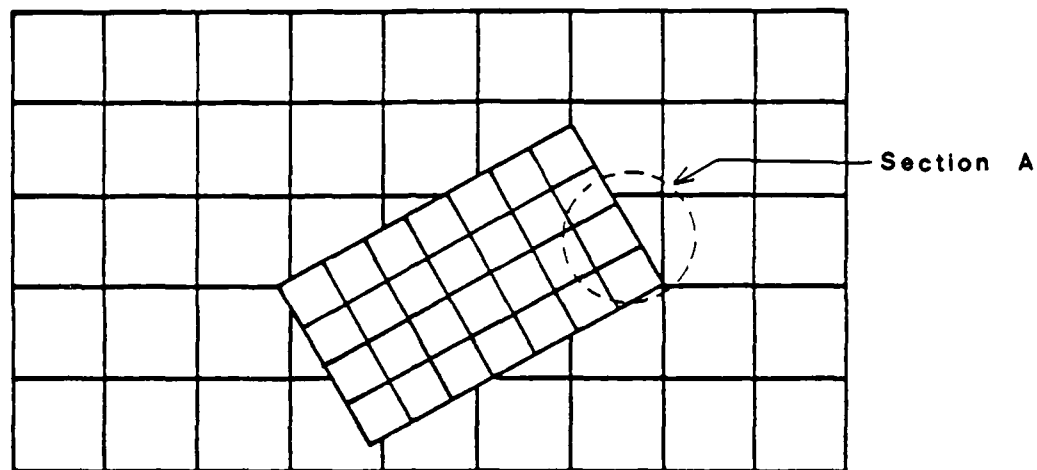
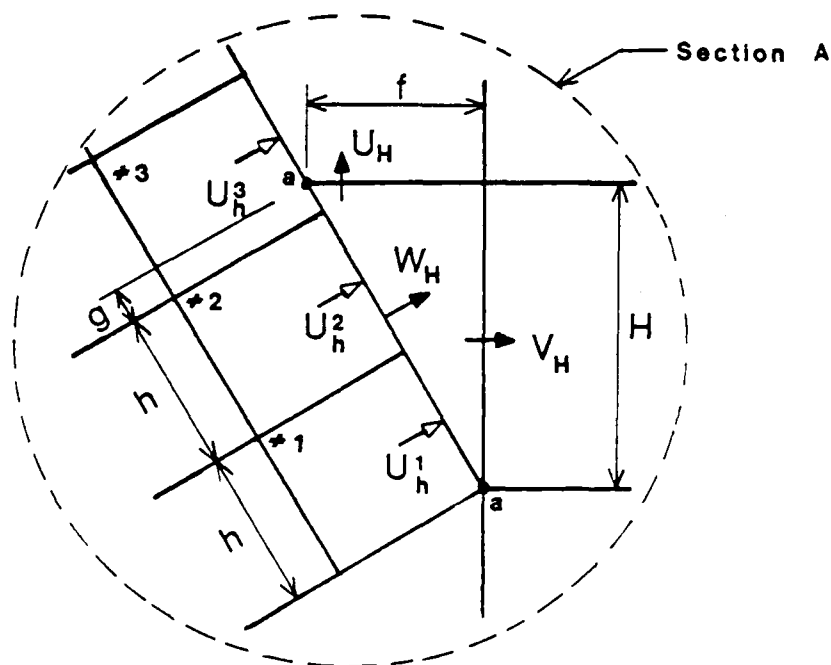


Fig. 7.3. Conservative coarse-to-fine grid interpolation.



(a)



(b)

Fig. 7.4. Interpolation for rotated grids.



Chapter 8

APPLICATION TO THE LAMINAR BACKSTEP

In this chapter, we present the results of adaptive computations of laminar flow over a backward-facing step. We first describe the physical flow and the corresponding computational model. The results of uniform grid calculations and a justification for using nonrotated refinement are then given. Next, results of preliminary adaptive grid calculations are presented. A performance evaluation at a single Reynolds number follows. Finally, we show results of adaptive calculations over a range of Reynolds numbers.

8.1 Description of the Problem

The flow through a straight channel having a sudden asymmetric expansion is called the backward-facing step problem (see Fig. 8.1a). Separated flows resulting from such changes in geometry are common in energy conversion devices; the device's performance often depends on the structure of the flow in these regions. As a result, the backstep problem has received considerable experimental and theoretical attention. In this section, we discuss the experimental flow which we shall simulate.

8.1.1 Experiment of Armaly et al.

Armaly et al. (1983) studied the flow through a two-dimensional sudden expansion, having an expansion ratio of 1:1.94. The test section had a long, straight inlet channel to provide a uniform inlet flow. Similarly, the exit channel was long to allow a fully developed velocity profile to develop.

Experiments were conducted over a range of Reynolds number covering the laminar, transitional, and turbulent flow regimes; we discuss only the laminar results. Streamwise velocity profiles were recorded, primarily in the section downstream of the backstep. The reattachment length, x_R , was measured as a function of the Reynolds number. Prediction of this parameter is difficult and is therefore used to check the accuracy of a numerical method.

The measured reattachment length is plotted in Figure 8.19 for Reynolds numbers, $50 < Re < 600$. The Reynolds number is defined:

$$Re = \frac{u_m h}{\nu}$$

where u_m is the maximum inlet velocity, h is the step height, and ν the kinematic viscosity. The flow is laminar in the range $50 < Re < 900$. Above $Re = 900$, the flow begins to undergo transition prior to reattachment. At still higher Reynolds numbers, the entire flow becomes turbulent.

The streamwise velocity in the inlet channel at the sudden expansion was measured and found to be "close to that of a fully developed channel flow, with a slight deviation from a parabolic profile." The normal velocity at this location was not measured. Fully developed conditions were found to occur in the outlet channel; the location moved downstream with increasing Reynolds number.

Cross-channel measurements were made to determine the two-dimensionality of the flow. The flow in the plane of the sudden expansion was found to be two-dimensional. The flow in the channel downstream of the step was also two-dimensional at low Reynolds numbers, but became three-dimensional for $Re > 300$, with 3-D effects increasing with Re . Additionally, an elongated recirculation bubble appears on the top wall for $Re > 300$. The location and length of this bubble is also a function of the Reynolds number.

Because the geometry is simple, the boundary conditions well-defined, and the data available, this flow is useful for evaluating the accuracy of a numerical method. We describe our computational model of this flow next.

8.1.2 Computational Model

The flow is simulated on a rectangular domain, using a uniform base grid shown in Fig. 8.1b. (The mesh size may be different in the two coordinate directions.) An expansion ratio of 1:2 is used so that the step-corner lies on a mesh point. The box length is fixed in the y-direction; the x-direction length, x_L , was variable. It was taken

to be four times the experimental reattachment length, as recommended by Armaly et al.

A parabolic streamwise velocity profile and zero normal velocity are specified at the inlet. Along all walls, both velocity components are zero. Fully developed velocity conditions, $\partial u / \partial x = \partial v / \partial x = 0$ are used at the outlet, $x = x_L$.

Note that two approximations are made in the model. First, the modeled expansion ratio is 3% larger than the experimental one. To a first approximation, computed reattachment lengths should be $\sim 3\%$ larger than the experimental values. Calculations were made by Zebib and Homsy (1984) on nonuniform grids using both expansion ratios. Their results confirm that the reattachment length is proportional to the step height.

The second approximation concerns the inlet velocity. Because the experimental streamwise velocity had a slight deviation from a fully developed profile, a small, nonzero normal velocity is expected in the plane of the expansion. Use of zero normal velocity in the computation will cause the computed reattachment lengths to deviate from experimental values, but the effect should be small. A better approach is to use an L-shaped computational domain and specify fully developed channel conditions farther upstream in the inlet channel.

As a result of these approximations, computed reattachment lengths should agree with the experimental data to within 10%, for Reynolds numbers for which the flow is two-dimensional, i.e., for $Re < 300$.

8.2 Uniform Grid Calculations

During the development of the adaptive program, uniform grid calculations were made for testing and debugging the basic solver. We also experimented with three convective-difference schemes and evaluated the error estimation procedure on uniform grids. In this section, we present results taken from these studies.

All calculations presented in this chapter were made on a VAX 11/780 in double precision.

8.2.1 Velocity Profiles

All adaptive calculations were made in the range $100 < Re < 600$. For these Reynolds numbers, all flowfields display similar behavior. Velocity profiles for $Re = 100$ with $x_L/h = 12$ are shown to illustrate the qualitative nature of the flow; the calculations are not described in detail. Quantitative results are given in sections that follow.

Figure 8.2 shows two surface views of u , the x-component of velocity. (These plots are somewhat distorted; the actual aspect ratio (y:x) of the domain is 1:6.) u varies smoothly from the parabolic profile to another fully developed profile at the outlet. The region of negative u behind the step indicates the recirculating region. Reattachment occurs where u changes sign along the bottom wall. For this calculation, $x_R/h \sim 3.6$. In the latter half of the channel, streamwise gradients are small.

Figure 8.3 gives v , the y-component of velocity; it has rapid variation near the sudden expansion. Downstream of the inlet, v is negative in the shear layer. Farther downstream, v drops towards zero. Immediately behind the step, v is positive in the recirculating region. The maximum magnitude of v is approximately 10% of the maximum magnitude of u .

Greater detail can be seen in Figs. 8.4 and 8.5, where u has been plotted along lines of constant y/h and x/h , respectively. The behavior of u near the bottom wall is seen in Fig. 8.4a. The smooth behavior of u in the vicinity of the step is shown in Figs. 8.4b-e. These figures also show the smallness of the streamwise gradients farther downstream. Specification of fully developed conditions at the outlet is consistent; no abrupt changes occur near this boundary.

Figure 8.5a shows the variation of u with y in a plane passing through the the recirculation region. In Fig. 8.5b, the u -profile just downstream of reattachment is plotted. Figures 8.5c,d show that u is nearly parabolic downstream; a reference parabola is included in these figures.

In Fig. 8.6, v is plotted at constant y/h locations. Figures 8.6b-d illustrate the rapid variation of v just downstream of the step. As discussed in Section 8.2.3 below, this region has the largest truncation error and thus requires the most grid refinement. Figure 8.7 illustrates the y -variation for v , which becomes more gradual as the flow moves downstream.

8.2.2 Comparison of Convective Difference Schemes

For $Re = 100$, mesh-refinement studies were made using central (CD) and Patankar's power law (HY) differencing schemes for the convective terms. (Both schemes are described in Section 6.3.) Table 8.1 summarizes the important parameters for the calculations. Initial guesses for each case were bootstrapped from the previous case.

Table 8.1 shows how the number of iterations for convergence depends on the total number of grid points; both schemes converged in approximately the same number of iterations, except for Case 4. Normalized cpu times were 0.007 and 0.009 sec/iteration/cell for HY and CD, respectively. The difference in costs is due to the calculation of the defect-corrections in the CD scheme (see Section 7.2). On a per iteration basis, the CD scheme is slightly more expensive, although, as discussed below, the payoff in accuracy is substantial.

The Case 5 CD calculation converges in relatively few iterations, because its solution is not very different from the Case 4 solution from which it was bootstrapped. From this near mesh independence, we concluded that the 192×128 CD calculation is a good approximation to the exact solution. The calculated x_R for this grid is approximately 10% larger than the experimental value, as expected (see Section 8.1.2).

Figures 8.8 and 8.9 are typical plots that illustrate the accuracy and convergence of the solution as the mesh size is decreased. They show that, on the same mesh, the CD solution error is significantly smaller than that for HY, and the CD error decreases faster as the mesh is refined.

Taking the 192×128 CD result as a good approximation to the exact solution, the rms solution errors were calculated for all other cases. These errors are tabulated in Table 8.2.

The errors for the 96 64 CD case are larger than expected. There are two causes for this. First, they are not accurate, as they are comparable in magnitude to the iteration-convergence criterion, 10^{-4} . Secondly, when the criterion is reduced to 10^{-5} , the errors on the 96 64 grid become approximately one-half of the respective errors on the 48 32 grid; the method becomes first-order accurate (see discussion below). Since the 96 64 and 192 128 solutions are nearly identical, we conclude that the error in the 96 64 solution is negligible at interior grid points and that the significant error is caused by first-order approximations, similar to (6.3.6) used at boundary cells (see section 6.3.2).

The errors are also plotted in Fig. 8.10. These plots also indicate the accuracy and convergence properties of the two methods. The order of accuracy of the method ("p" in Eq. (3.7.1)) is the slope of the log plot. We find that HY is first order ($p = 1$), CD second order ($p = 2$) as expected.

To summarize, we have shown that central differencing for the convective terms is more accurate than Patankar's power-law scheme. The added expense of its implementation (through the defect-correction method) is small, and the correction procedure does not degrade the overall convergence rate of the method.

As indicated in Table 8.1, central difference solutions were calculated for cell Reynolds numbers as high as 100. No stability problems were encountered and no wiggles were found in solutions; clearly, the commonly applied condition, $Re_x < 2$, is too restrictive.

Only a few calculations were made using the QUICK scheme (described in Section 6.3). One calculation was made at $Re = 100$, using the Case 2 grid in Table 8.1. The method converged in 76 iterations, giving $x_R/h = 3.85$, similar to the CD calculation, and the rms errors were somewhat smaller. The cpu time was 0.009 sec/iteration/cell. For this case, the performance of QUICK is comparable to CD.

Two QUICK calculations were made at $Re = 600$. The first was on a 42 16 grid with $x_L/h = 42$. Convergence was achieved in 900 iterations, and oscillations were present in the resulting solution. A second calculation was attempted on an 84 16 grid, with the same x_L/h .

However, after 1000 iterations, the run was aborted, since the solution was converging very slowly. This can be compared with a 420×20 CD calculation at the same Re , that converged in 1400 iterations. Because of the slow convergence at high Re , QUICK was dropped from consideration. (Although it was not tried, QUICK implemented by the defect-correction scheme may converge faster.)

Due to its accuracy, stable implementation, and wiggle-free solutions at large Re , central differencing was selected for use in all adaptive calculations presented in Sections 8.4 and 8.5 below.

8.2.3 Exact vs. Estimated Errors

Uniform grid calculations were also made to test and evaluate the error-estimation method. In this section, we show the results of calculations made at $Re = 100$, using central differencing.

Calculations were made using 24×16 and 12×8 grids. The solutions were inserted into (7.4.1), with $p = 2$ to give the u -solution error estimate; the v -error estimate was similarly computed. To evaluate the accuracy of the estimates, the exact solution error ($u_h - u_{\text{exact}}$) was computed using the 192×128 central difference solution as the "exact" solution.

Absolute values of the errors are plotted in both contour and surface views. When considering these figures, recall that the actual aspect ratio ($y:x$) for the rectangles is 1:6; in the plots, the ratio appears to be 1:1.5. The surface views provide a perspective of the errors. The plotted values have been normalized by the maximum magnitude so the differences in the elevations are exaggerated.

The estimated and exact solution errors for u and v are given in Figs. 8.11 and 8.12, respectively. The plots show that the topography of the exact error is well represented by the estimate, especially when the coarseness of the grid is considered. Away from the step, the estimated magnitudes are relatively accurate, and are generally high. For example, the location and magnitude of the maximum error in u is predicted well. The heights of the humps in the v -error are similarly well predicted. Overestimation of the error is preferable to underestimation.

Near the corner, the error estimate is inaccurate because the higher-order terms in the expansion (3.7.1) are large. Because the h - and $2h$ -solutions differ considerably in this region, the error is poorly predicted. The accuracy of the estimate improves on finer grids.

The truncation error estimate is computed by substituting the solution and its solution error estimate into (7.4.2). The exact truncation error is evaluated by substituting the 192×128 central difference solution into the difference operator in (7.4.2).

The truncation error for u and v are plotted respectively in Figs. 8.13 and 8.14. As discussed in Section 3.7, these estimates are not expected to be as accurate as the solution error estimates. Although there is less similarity between exact and estimated topographies for these errors, the estimates do indicate where the truncation error is large and, thus, where grid refinement is required. Truncation error also is generally overpredicted. These plots illustrate the "noisy" behavior of the truncation error. Smoothing of the estimates is required if they are to be used in a global refinement method.

These results indicate that the Richardson error procedure provides reasonably accurate estimates of the solution and truncation errors, even on coarse grids.

Before closing this section, two final points are discussed. We first discuss the relationship between solution error and truncation error and some of the implications. We then give a justification for using nonrotated grid refinement.

Consider the contour plots of the errors for both u and v , Figs. 8.11-8.14. The figures illustrate how the truncation error is convected and diffused through the flowfield, resulting in the solution error, as discussed in Section 3.7. Since the truncation error has a long-range influence on the solution error, the active solution method is used for adaptive computations of this flow. The truncation error shows that refinement is required primarily in the vicinity of the sudden expansion.

These error plots provide a justification for using nonrotated refined grids. The refinement region defined by the truncation error

estimates for u and v encloses most of the upstream portion of the base grid.

Preliminary adaptive calculations showed that a single rectangle with a small angle of rotation ($< 10^\circ$ relative to the x -axis) is generated, with its corners falling outside of the problem domain, as indicated in Fig. 8.15a. The rectangle would have to be reduced in size and unenclosed bad points fit with overlapping boundary-aligned grids, similar to the procedure described in Chapter 5. The rectangles that would result from this procedure are sketched in Fig. 8.15b.

The cost of generating the solution would increase due the work required for solving on overlapping grids at each level. Since the rotation is small, this added cost is not expected to outweigh the benefit of reduced "rotational error". As a consequence of this, and also due to the difficulty of constructing a conservative 2-D interpolation scheme, we elected to use boundary-aligned grid refinement for the backstep problem.

8.3 Preliminary Adaptive Calculations at $Re = 100$

Preliminary adaptive calculations were made during the development and testing of the program; results of these studies are presented in this section. A notation for the grids is first given, followed by results of studies that investigated the convergence properties and iteration strategy for the active solution method. (The active solution procedure is described in detail for a typical calculation in Section 8.3.2.) Finally, results are given to illustrate the influence that the location of "fictitious" internal, fine-grid boundaries has on the solution accuracy.

All calculations in this section were made at $Re = 100$. Since the calculations were only preliminary, Patankar's power-law convective difference scheme was used; central differencing is expected to give similar results.

8.3.1 Notation for Refined Grids

In the adaptive calculations described in the remainder of this chapter, the geometry of all grids is similar. In this section, a simplified notation for the grids is defined.

Refined grids are boundary-aligned, as discussed in Section 8.2.3. Each base or refined grid, G_k covers a rectangular domain: $x \in (0, x_{L_k})$; $y \in (0, 2)$ as indicated in Fig. 8.16. The coordinates (x, y) are normalized by the step height, h . Grids are described using the notation:

$$G_k: N_x \times N_y ; x_{L_k}$$

where N_x and N_y are the number of cells in the x - and y -directions, respectively. The domain of G_k is denoted Ω_k .

The internal, fine-grid boundary for a refined grid lies along the line $(x_{L_k}, 0 \leq y \leq 2)$ and is denoted γ_k .

8.3.2 Convergence of the Active Solution Method

A calculation was made using a two-level grid system to investigate the convergence properties of the active solution method. The parameters for this calculation are:

$$G_0: 12 \times 8; x_{L_0} = 12$$

$$G_1: 12 \times 16; x_{L_1} = 6$$

Navier-Stokes (inner iteration) convergence criterion:

$$\max_{i,j} |\underline{u}^n - \underline{u}^{n-1}| < 10^{-4} \quad (8.3.1)$$

Active solution (outer iteration) convergence criterion:

$$\max_{\gamma_k} |\underline{u}^m - \underline{u}^{m-1}| < 10^{-4} \quad (8.3.2)$$

where n is the index for inner iterations, m for outer iterations.

A converged solution is first calculated on G_0 . Then for G_1 , exact boundary conditions are applied at all problem boundaries. Along the internal boundary, γ_1 , the normal velocity component, u , is interpolated from G_0 using the conservative interpolation procedure described in Section 7.3.1. The tangential component, v , is bilinearly interpolated. A converged solution is then calculated on G_1 .

Next, correction terms are calculated for G_0 , as described in Section 7.2. For G_0 boundary points coincident with G_1 's boundary, the solution is linearly interpolated from G_1 's solution. Modification of these boundary conditions is required to conserve mass on G_0 , as discussed in Section 7.3.2. (The modification needs to be done only once, during the first active iteration on a newly created grid system.) A converged solution is then calculated on G_0 . This completes the first active iteration. The process is continued until the inner and outer convergence criteria (8.3.1)-(8.3.2) are satisfied at all grid points.

Figure 8.17 plots e^m , the maximum change in the boundary values along γ_1 versus iteration number. For example, for u , e^m is defined as:

$$e_u^m = \max_{\gamma_1} |u^m - u^{m-1}| \quad (8.3.3)$$

The change in v is also plotted in the figure. The method is seen to converge linearly, and the convergence is achieved in nine outer iterations. As discussed in Section 3.6, monotonic linear convergence suggests that SOR, for example, can be used to increase the convergence rate. We used an alternative approach described in the next section.

8.3.3 Iteration Strategy: Inner vs. Outer

Because the active scheme is iterative, the solution on a given grid need not be fully converged before switching to another grid. This is illustrated in Fig. 8.17, where it is seen that the change in the solution along the fine-grid boundary is larger than the inner convergence criteria (8.3.1) for most of the outer iterations. At the m -th outer iteration, the work required to converge the solution to an error much less than e^{m+1} is wasted.

The optimum iteration strategy is to have the inner convergence error during the m^{th} outer iteration be somewhat smaller than e^{m+1} . If the convergence is linear, e^{m+1} can be predicted from e^m and e^{m-1} , and then reduced by an appropriate factor to provide an inner convergence criterion.

An alternative approach is to iterate a fixed number of times on each grid, before switching. This is done in multigrid methods, in which 2-4 inner iterations are typically made on each grid. Using the following two-level grid system:

$$G_0: 24 \times 16; \quad x_{L_0} = 12$$

$$G_1: 24 \times 32; \quad x_{L_1} = 6$$

active solutions were made with a fixed number of inner iterations on each grid, and also with the inner convergence criterion (8.3.1). The active convergence criterion was (8.3.2) for all cases. The work to calculate each converged active solution was estimated by evaluating the number of times a grid point was swept, and then summing over all points in both grids. This was compared to the work required to calculate a solution on a grid having the same mesh sizes as G_1 . The results are given in Table 8.3.

Case 1 in Table 8.3 shows that fully converging on each grid requires more work than calculating on a uniform fine grid. For this calculation, five iterations is optimum. We used five inner iterations for the majority of adaptive calculations. However, the inner iteration strategy had to be modified at higher Reynolds number, as discussed in Section 8.5.

8.3.4 Effect of Internal Fine-Grid Boundary Location

Before closing this section, we show results of calculations which illustrate the effect that the location of the internal fine-grid boundary has on the solution accuracy.

Two two-level grid systems were used. The base grid, G_0 was the same in both cases. The refined grids, G_{1a} and G_{1b} , differed only in the location of their downstream boundaries. The grids are:

$$G_0: 24 \times 16; x_{L_0} = 12$$

$$G_{1a}: 16 \times 32; x_{L_{1a}} = 4$$

$$G_{1b}: 24 \times 32; x_{L_{1b}} = 6$$

Converged active solutions were calculated on both grid systems. A solution was also calculated on a uniform fine grid having the same mesh sizes as the G_1 grids. A typical plot of v at a constant y/h is given in Fig. 8.18.

Figure 8.18a shows the case with $x_{L_1} = 4$; the error in the active solution is larger than the case with $x_{L_1} = 6$ shown in Fig. 8.18b. The solution error results from both local truncation error and interpolation error at the fine-grid boundary. If a higher-order interpolation scheme were used, the solution error should be reduced, allowing the boundary x_{L_1} to be placed farther upstream than say $x = 6$. As a result, the adaptive method would be more efficient, since the refined area would be reduced.

8.4 Adaptive Performance Evaluation for $Re = 100$

An adaptive calculation was made at $Re = 100$, using central differencing. The results are compared to the central-difference, uniform-grid calculations described in Section 8.2.2, to evaluate the performance of the adaptive method. The results of the comparison are discussed in this section.

The inner and outer iteration criteria used in the calculation are (8.3.1-2); the maximum allowed estimated truncation error, $\tau_{\max} = 2.5 \times 10^{-4}$. Five inner iterations are made on each grid for active calculations. Tables 8.4 and 8.5 summarize the problem parameters, along with the adaptive results.

The resulting two- and three-level active solutions were compared to the central difference, uniform grid solutions having the same mesh sizes as the finest level grid. (The two-level solution was compared to the 48×32 grid, the three level solution to the 96×64 grid; both grids are summarized in Table 8.1.)

Rms errors for the adapted solutions were calculated relative to the 192×128 CD solution as done earlier, and are given in Table 8.4. The table shows that the accuracy in the adapted solution for each level of refinement is nearly the same as the accuracies in the respective uniform-grid solutions.

The ratio of cpu time for adaptive calculations to the cpu time for uniform-grid calculations is also given in Table 8.4. For the adaptive calculations, the cpu time includes the time required to generate the solution at the previous level.

The adaptive method is 40% faster than the uniform-grid calculation with one refinement level. The three-level adapted solution is approximately six times faster than the corresponding uniform-grid calculation.

The backstep flow poses a severe test for our adaptive method, since a large part of the problem domain needs to be refined. The efficiency of the method will be larger for problems having smaller, more localized refinement regions.

To summarize, we have demonstrated that an active, adaptive calculation of the laminar backstep flow at $Re = 100$ has the same accuracy as a uniform fine-grid calculation but is six times faster.

8.5 Adaptive Results for $100 < Re < 600$

In this section, we present the results of adaptive calculations made at higher Reynolds numbers, and compare the computed reattachment lengths against the experimental values. The expense of calculations at higher Reynolds numbers prohibits calculating uniform fine-grid solutions to compare against the adaptive ones. As shown below, the adaptive calculation at $Re = 600$ pushes the limits of the VAX 11/780.

Adaptive calculations were made at $Re = 100, 200, 300, 450$ and 600 , using central differencing. The base and resulting refined grids for each calculation, along with problem parameters are summarized in Table 8.5.

For all calculations, the inner and outer convergence criteria are (8.3.1.,.2) as before. The maximum allowed estimated truncation error for each case is indicated in Table 8.5. Adaptive calculations were

stopped when the computed reattachment lengths had converged to within 1%; consequently, two refinement levels were required for all Reynolds numbers.

The adaptive calculations for $Re = 200$ and 300 were performed in the same manner as for $Re = 100$. The refinement regions for $Re = 100$ - 300 are similar. The computed reattachment lengths agree well with the experimental values. Note in Table 8.5 how the number of outer iterations for convergence, along with the cpu times, increase for these three cases.

At higher Reynolds numbers, the behavior of the flow solver SIMPLER required modification of the inner iteration strategy. On a single grid, for $Re > 300$, the convergence of SIMPLER was no longer monotonic (i.e., the change in velocity $(\underline{u}^n - \underline{u}^{n-1})$ for one iteration was not always smaller than that for the previous iteration.) Increasing the underrelaxation did not help.

Also, the intermediate solutions have spatial oscillations (or waves) which move through the grid and die out very slowly. The oscillations arise after only a few iterations. Consequently, if they are not removed before switching grids in an active calculation, the method causes them to be passed to all other grids in the system, with subsequent divergence of the solution.

The problem was remedied by iterating to a partial convergence, and thus removing the oscillations before switching to another grid. For $Re = 450$ and 600 , the following active iteration strategy was used. Two-level calculations were first converged to the inner and outer convergence criterion 10^{-3} , before reducing the criterion to 10^{-4} and fully converging the calculations. The three-level solutions used the 10^{-4} criterion. (This strategy could be used at all Reynolds numbers; further study is required to determine the best approach.)

The resulting refined regions are similar to the lower Reynolds number cases; however, the computed reattachment lengths are smaller than the experimental values. The cpu times also become very large.

As discussed in Section 8.1.1 above, Armaly et al. (1983) observed an elongated, secondary recirculation bubble on the top wall for $Re >$

300. At $Re = 450$ and 600 , the bubble is located approximately between:

$$8.5 < x/h < 15.5 \quad \text{for } Re = 450$$

$$11.5 < x/h < 19 \quad \text{for } Re = 600$$

This recirculation region also appears in our calculations and is located between:

$$9.5 < x/h < 15.5 \quad \text{for } Re = 450$$

$$10 < x/h < 20 \quad \text{for } Re = 600$$

Since the physical flow is three-dimensional at these Reynolds numbers, better agreement is not expected.

Computed reattachment lengths are plotted versus Reynolds number, along with the experimental data in Fig. 8.19. Also plotted are the calculations of Kim and Moin (1985), who also used central differencing but a different solution method. Their computational model is identical to ours (described in Section 8.1.2), and all calculations were made on a uniform, 100×100 grid, with $x_L/h = 30$.

Our results agree with those of Kim and Moin at all Reynolds numbers. (However, we cannot draw any conclusions regarding the global agreement between their solutions and ours.) The agreement with the experimental data is good for $Re < 300$. As discussed in Section 8.1.1, the experimental flow became three-dimensional at higher Reynolds number; this is responsible for the disagreement with the 2-D calculations.

In summary, we have made adaptive calculations of the laminar back-step flow over a range of Reynolds numbers. Calculations become quite expensive at higher Reynolds numbers; uniform fine-grid calculations would be prohibitively expensive. The results agree with a similar set of central difference calculations, and also with experimental data, in the range of Reynolds number at which the flow is 2-D.

TABLE B.1
PARAMETERS FOR UNIFORM GRID CALCULATIONS

Re = 100 ; $X_L = 12$

CASE	Nx * Ny	$\Delta x/h$	$\Delta y/h$	$\max Re_{\Delta x} $	(1) ITERATIONS		(2) XR/H	
					CD	HV	CD	HV
1	12 x 8	1.0	0.25	100	53	49	3.52	3.17
2	24 x 16	0.5	0.125	50	75	78	3.83	3.58
3	48 x 32	0.25	0.0625	25	106	~100	3.91	3.76
4	96 x 64	0.125	0.03125	12.5	232	~600	3.94	3.88
5	192 x 128	0.0625	0.015625	6.3	150	---	3.97	---

(1) CONVERGENCE CRITERIA : $\max \left(\left| u^n - u^{n-1} \right|, \left| v^n - v^{n-1} \right| \right) < 10^{-4}$

(2) EXPERIMENTAL $x_r \sim 3.6$

TABLE 8.2
UNIFORM GRID ERROR*

Nx * Ny	u e rns		v e rns	
	CD**	HY**	CD**	HY**
12 x 8	.0367	.0617	.0084	.0150
24 x 16	.0084	.0299	.0018	.0082
48 x 32	.0019	.0135	.0005	.0041
96 x 64	.0013	.0056	.0003	.0018

* RELATIVE TO 192 X 128 CD CALCULATION

** CD = CENTRAL DIFFERENCE
HY = PATANKAR POWER-LAW SCHEME

TABLE B.3
INNER ITERATION STRATEGY

CASE	INNER ITERATIONS	# OUTER ITERATIONS FOR CONVERGENCE	<u>WORK (ACTIVE)</u> WORK (UNIFORM)
1	Variable, used inner convergence criteria (8.3.1)	9	1.7
2	10	10	0.8
3	5	12	0.7
4	2	>30	>1

TABLE 8.4
ADAPTIVE VS. UNIFORM GRID PERFORMANCE FOR $Re = 100$

UNIFORM* GRID	RELATIVE LEVELS	u e r m s		v e r m s		$\frac{\text{WORK (adaptive)}}{\text{WORK (uniform)}}$
		UNIFORM	ADAPTIVE	UNIFORM	ADAPTIVE	
24 x 16	0	.0084	.0084	.0018	.0018	1.0
48 x 32	1	.0019	.0022	.0005	.0005	1.9
96 x 64	2	.0013	.0016	.0003	.0004	5.9

TABLE B.5
SUMMARY OF ADAPTIVE BACKSTEP CALCULATIONS

Re	X _R DATA	LEVEL	N _x * N _y	X _L	X _R	J _{max}	W*	INNER ITERATIONS	OUTER ITERATION		CPU HOURS
									1h	2h	
100	3.8	0	24 X 16	12.00	3.83	.00025	.85	75	--	--	.07
		1	26 X 32	6.50	3.91			5	12	7	.20
		2	6 X 64	0.75	3.92			5	8	--	.30
200	6.0	0	40 X 16	20.00	6.48	.00025	.85	281	--	--	.50
		1	40 X 32	10.00	6.51			5	20	10	.70
		2	6 X 64	0.75	6.51			5	30	--	2.30
300	8.6	0	208 X 16	26.00	8.79	.00010	.80	557	--	--	4.30
		1	142 X 32	8.90	8.79			5	48	16	5.70
		2	6 X 64	0.20	8.76			5	60	--	10.10
450	11.5	0	272 X 16	34.00	11.00	.00010	.60	371	--	--	4.40
		1	272 X 32	17.00	10.84			VARIABLE	14	9	57.00
		2	6 X 64	0.20	10.84			VARIABLE	3	--	13.00
600	14.3	0	240 X 16	30.00	10.81	.00025	.60	1049	--	--	10.00
		1	296 X 16	18.50	12.15			VARIABLE	24	6	200.00
		2	6 X 64	0.20	12.11			VARIABLE	45	--	100.00

*RELAXATION FACTOR IN EQUATION (6.4.11)

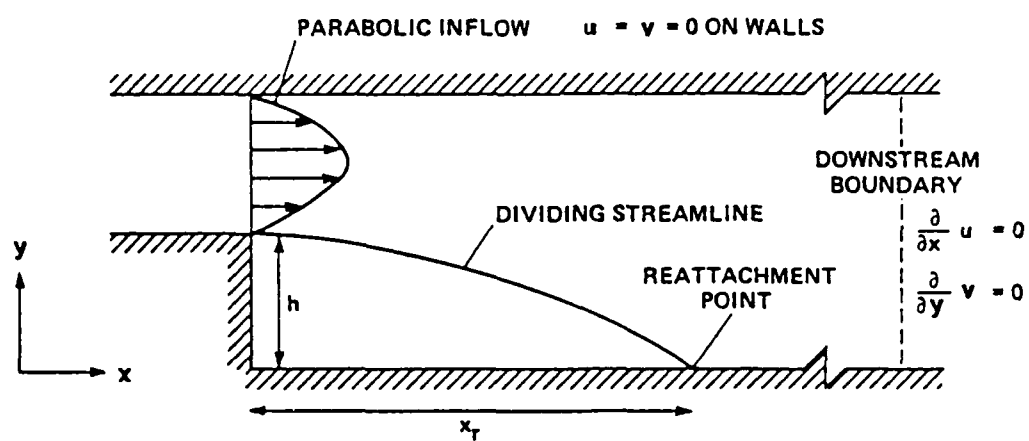


Fig. 8.1(a). Backstep geometry.

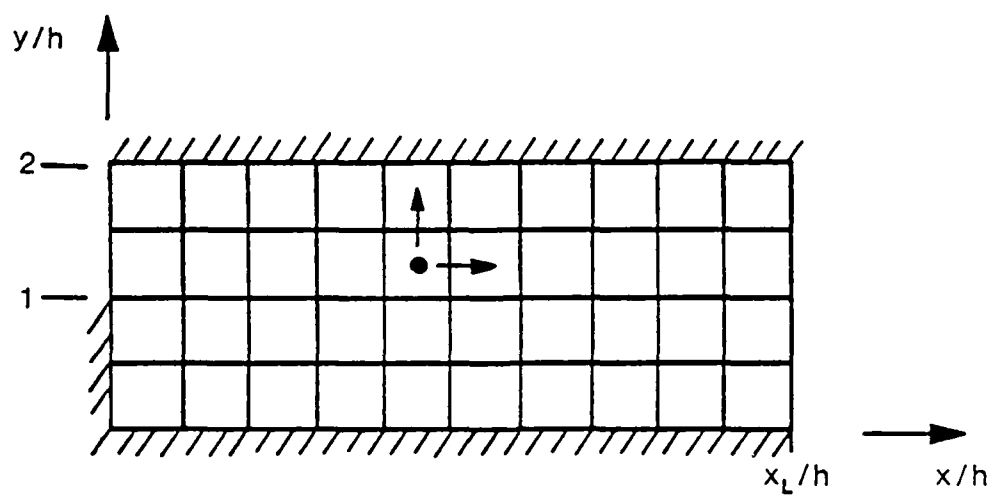
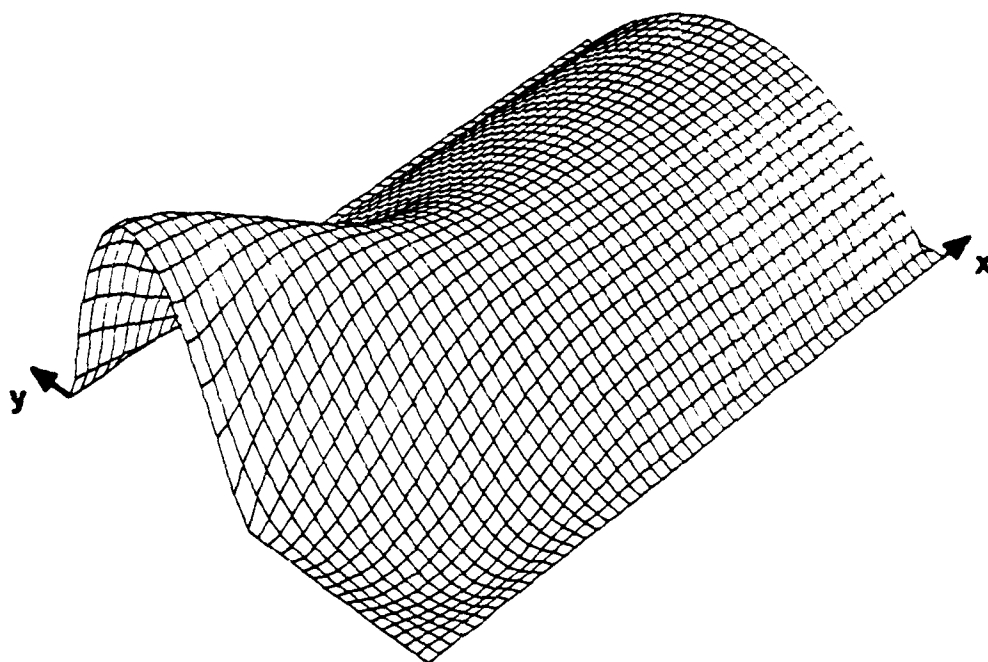
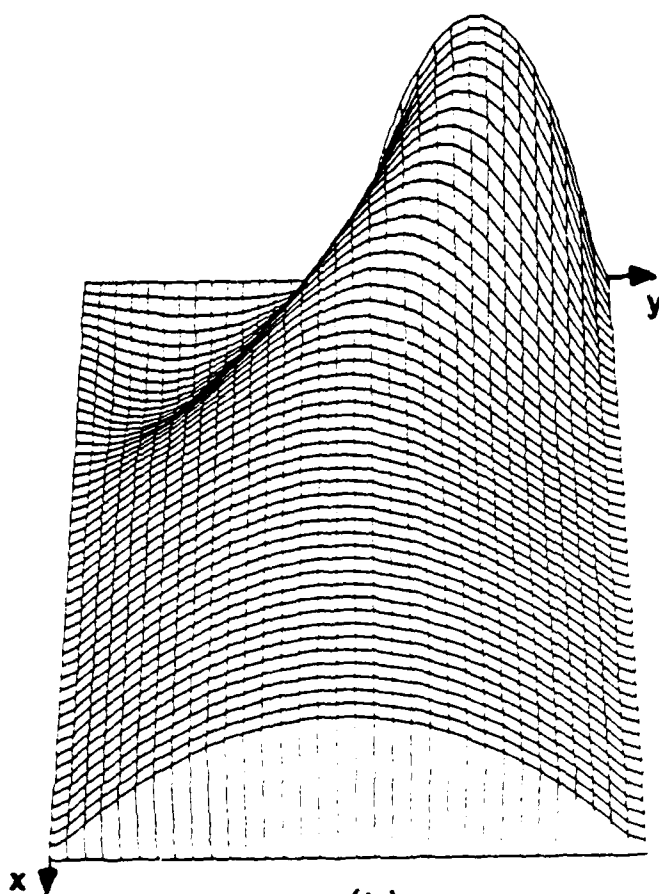


Fig. 8.1(b). Base grid for backstep.



(a)



(b)

Fig. 8.2. $u(x, y)$ for $Re = 100$.

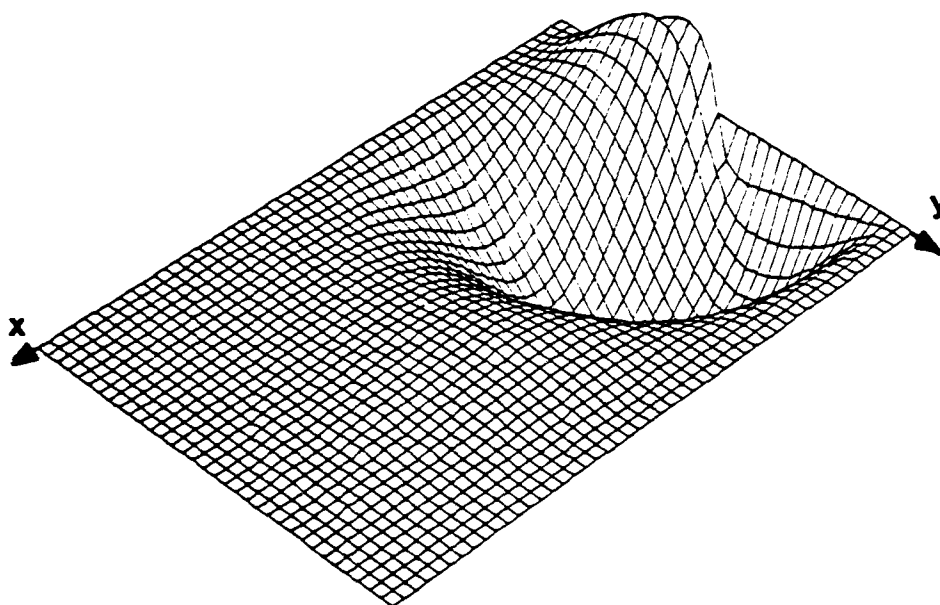


Fig. 8.3. $v(x,y)$ for $Re = 100$.

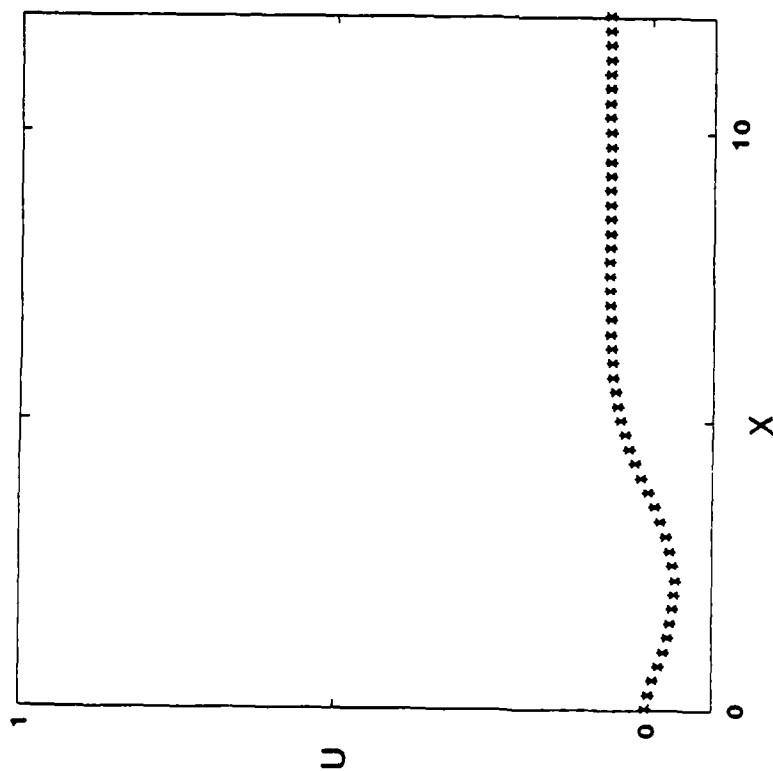


Fig. 8.4(a). $u(x, y = 0.0625)$ for $Re = 100$.

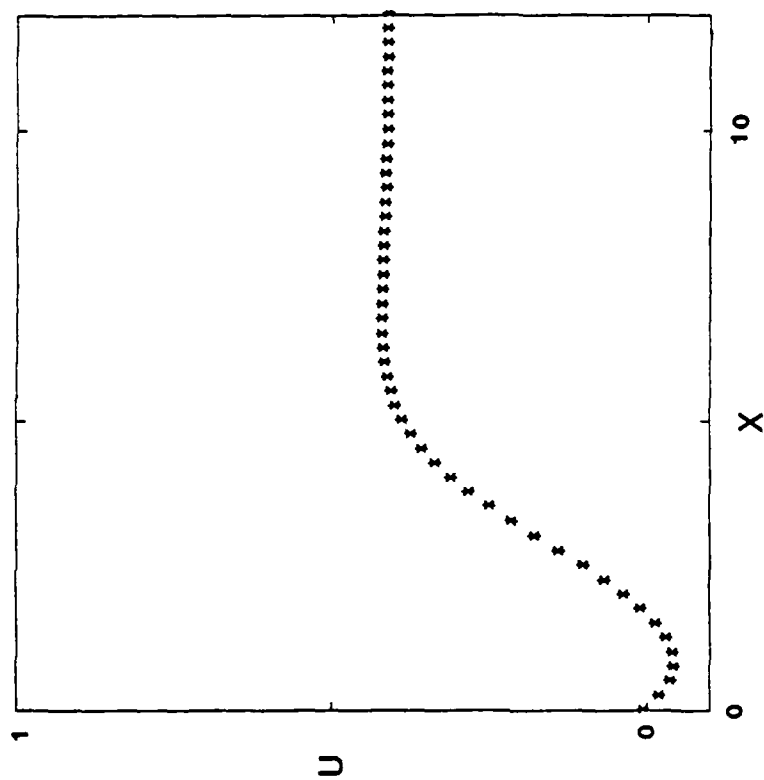


Fig. 8.4(b). $u(x, y = 0.5625)$ for $Re = 100$.

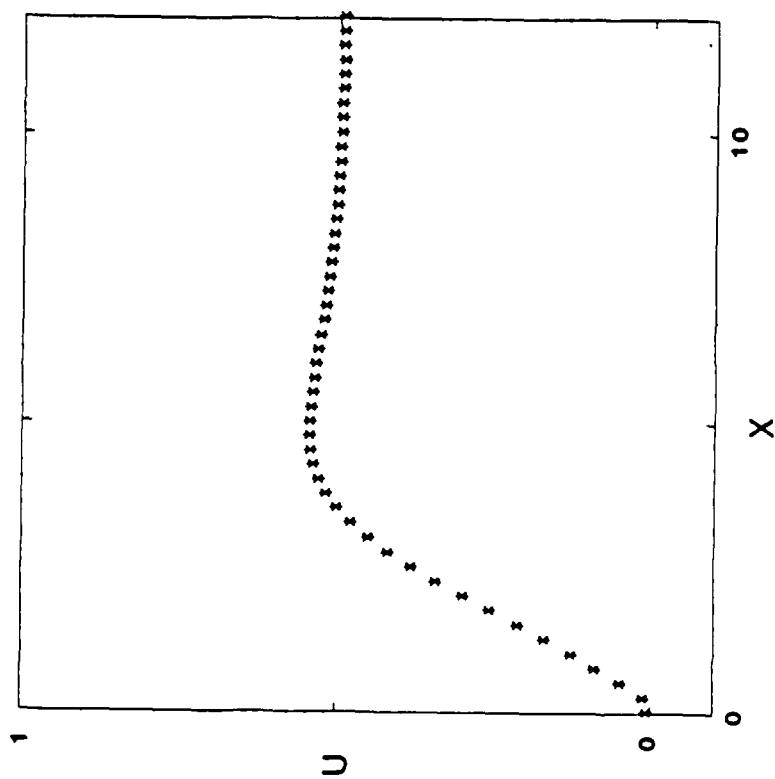


Fig. 8.4(c). $u(x, y = 0.8125)$ for $Re = 100$.

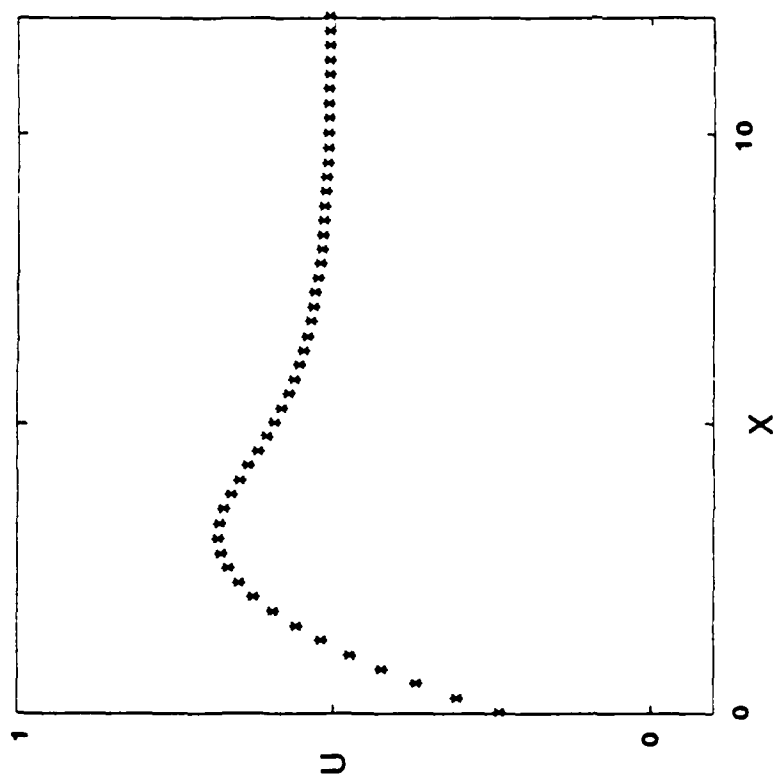


Fig. 8.4(d). $u(x, y = 1.0625)$ for $Re = 100$.

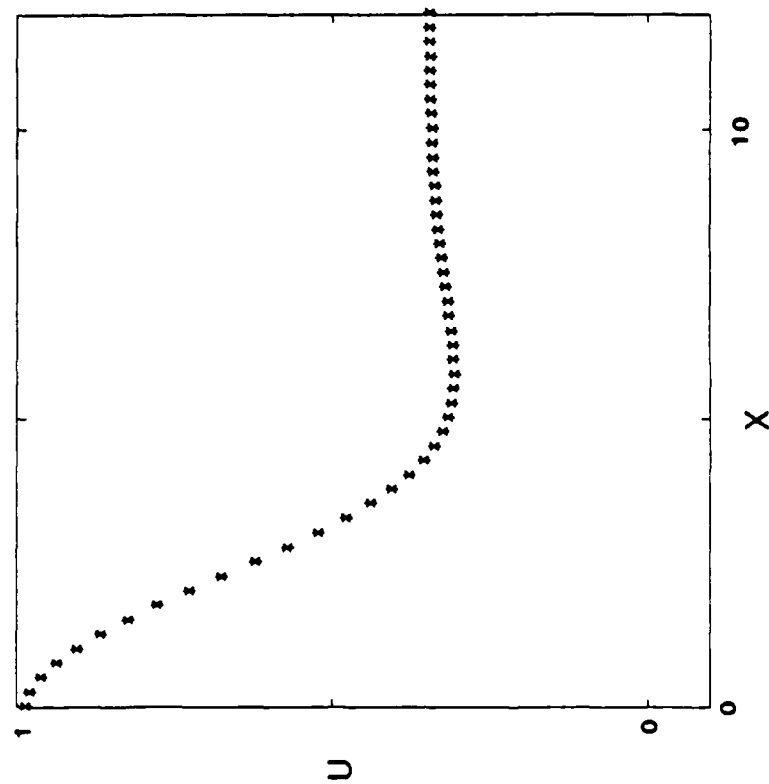


Fig. 8.4(e). $u(x, y = 1.5625)$ for $Re = 100$.

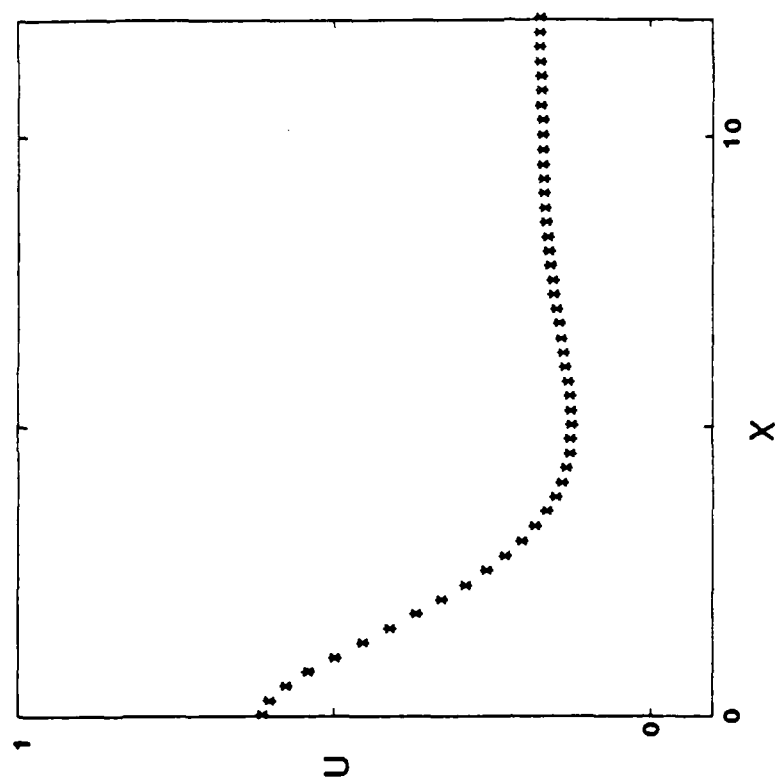


Fig. 8.4(f). $u(x, y = 1.8125)$ for $Re = 100$.

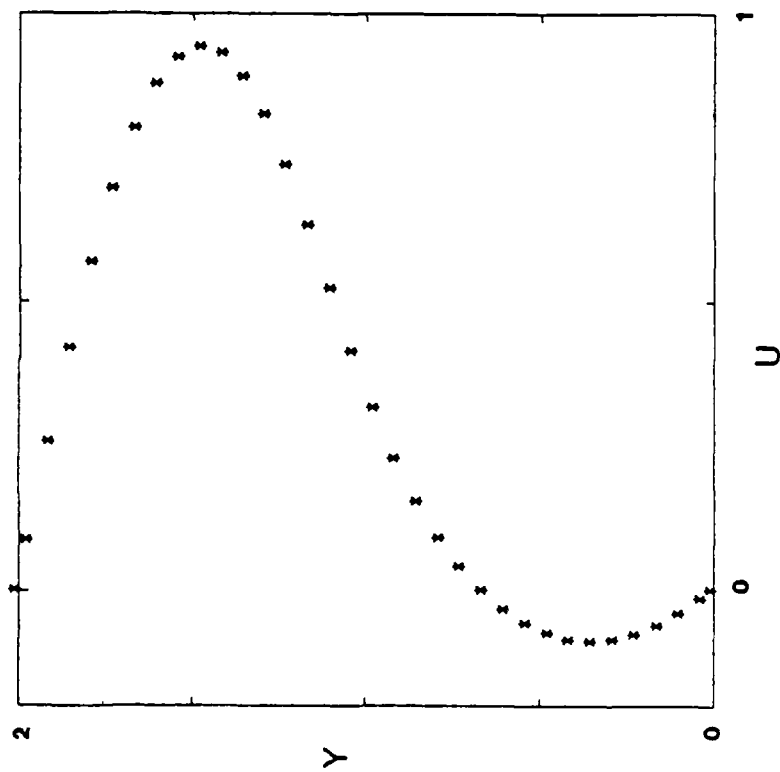


Fig. 8.5(a). $u(x = 1.0, y)$ for $Re = 100$.

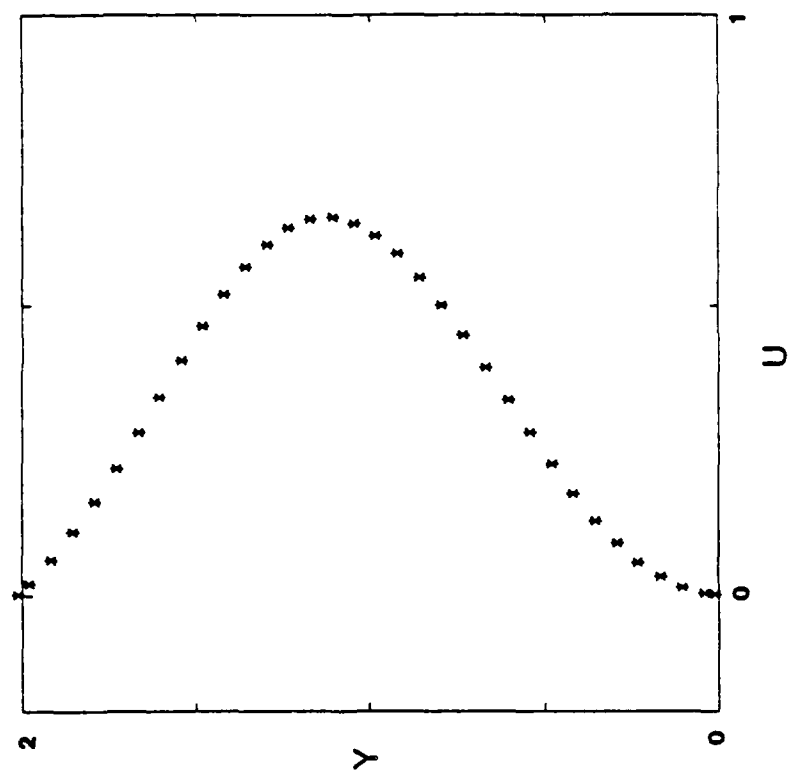


Fig. 8.5(b). $u(x = 4.0, y)$ for $Re = 100$.

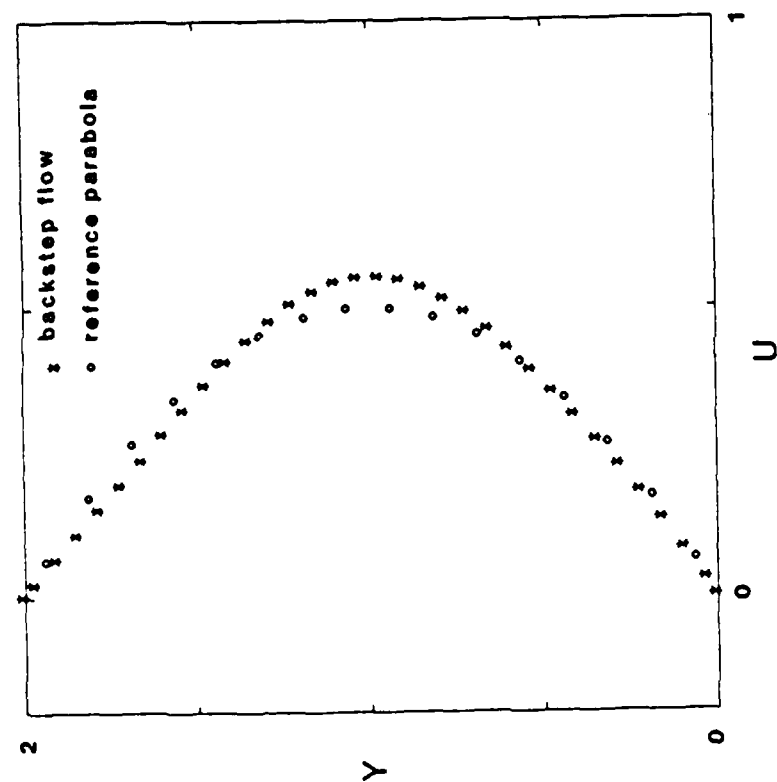


Fig. 8.5(c). $u(x = 6.0, y)$ for $Re = 100$.

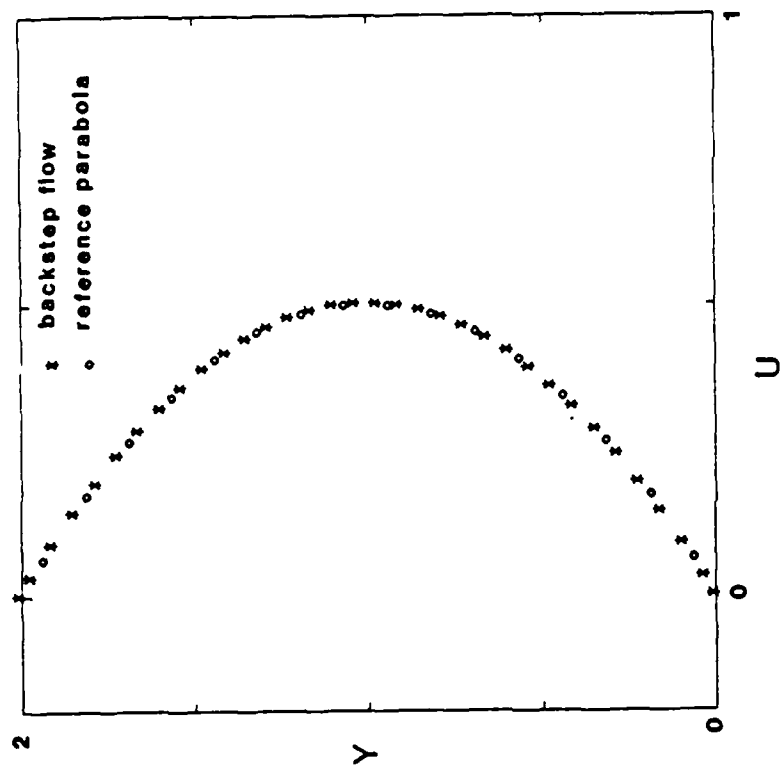


Fig. 8.5(d). $u(x = 12.0, y)$ for $Re = 100$.

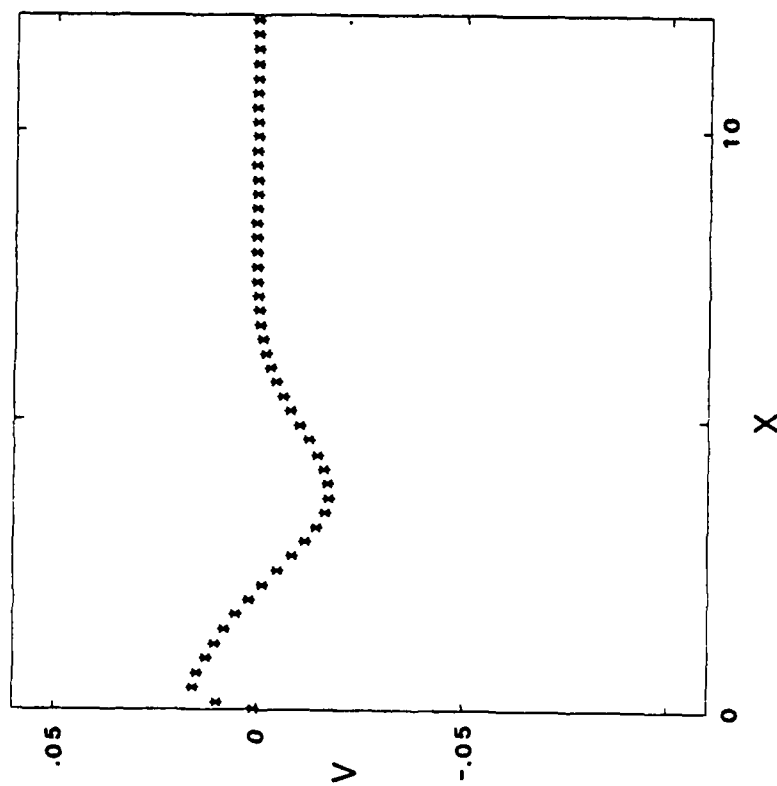


Fig. 8.6(a). $v(x, y = 0.25)$ for $Re = 100$.

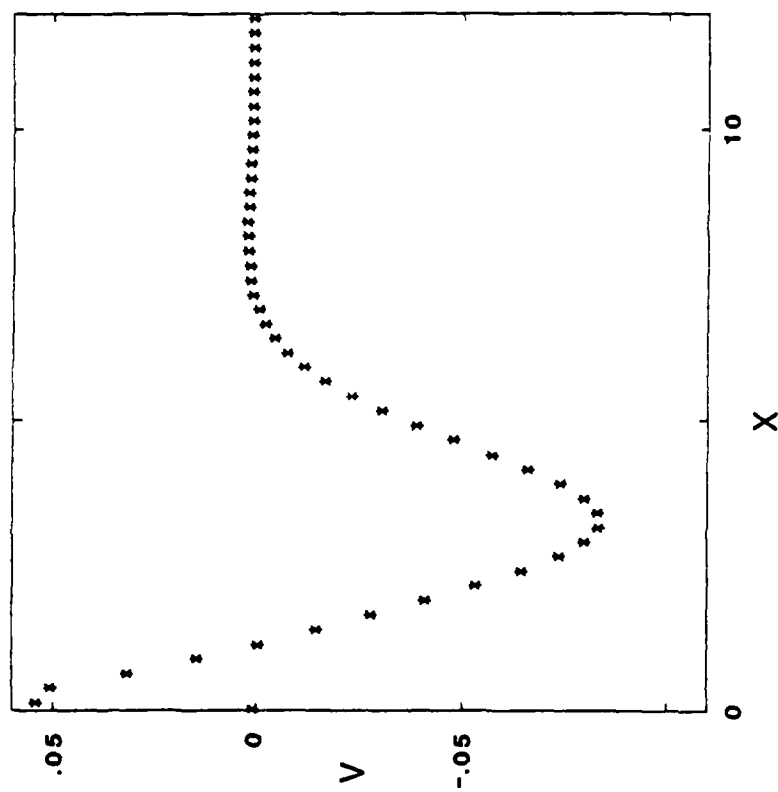


Fig. 8.6(b). $v(x, y = 0.75)$ for $Re = 100$.

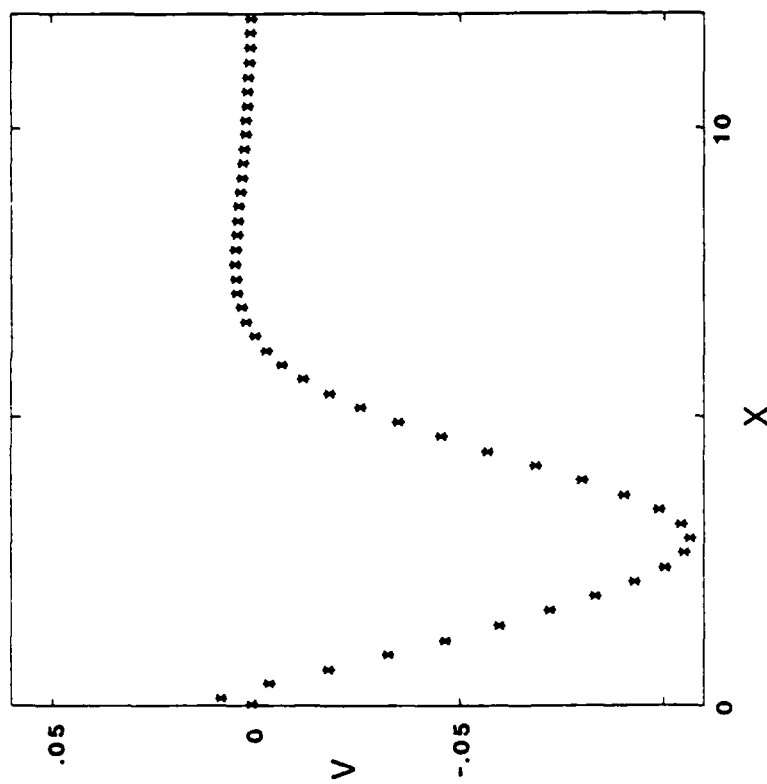


Fig. 8.6(c). $v(x, y = 1.0)$ for $Re = 100$.

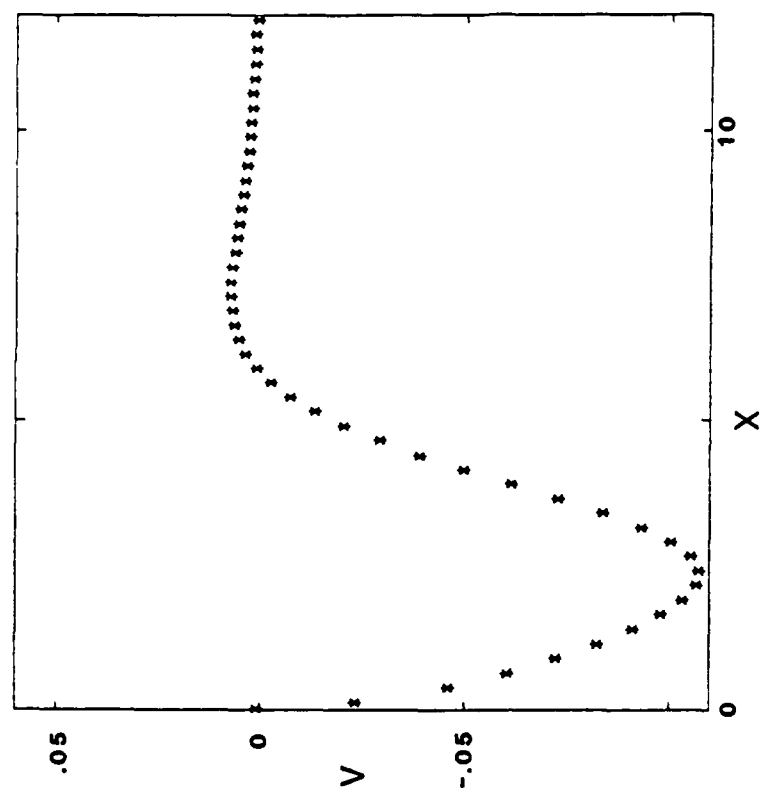


Fig. 8.6(d). $v(x, y = 1.25)$ for $Re = 100$.

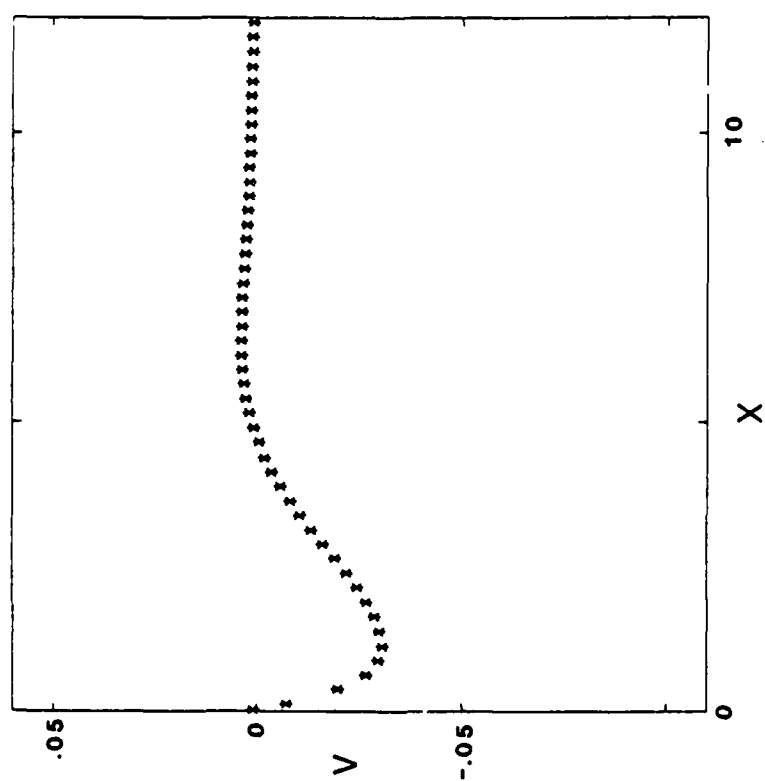


Fig. 8.6(e). $v(x,y = 1.75)$ for $Re = 100$.

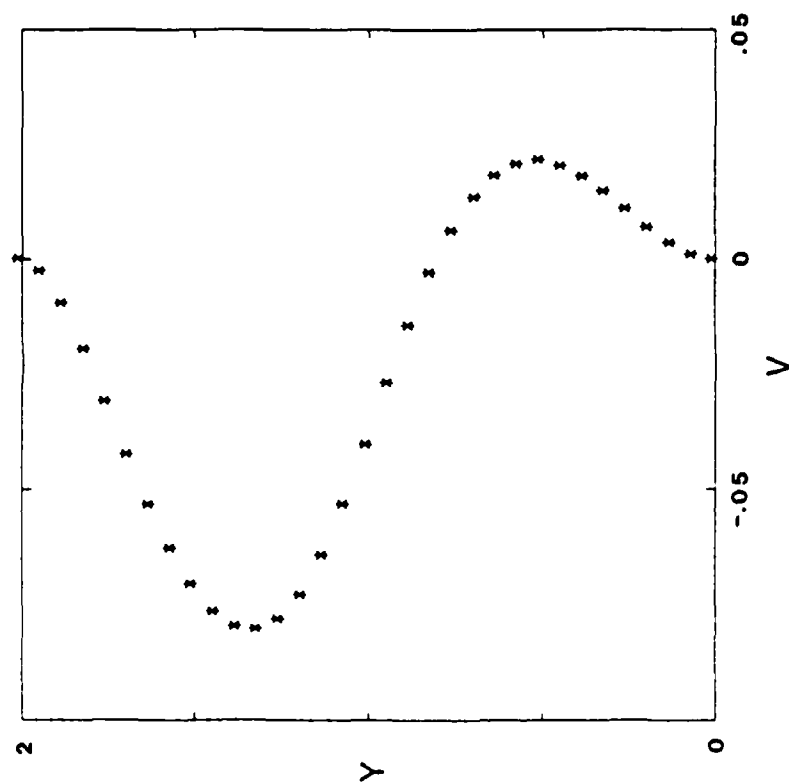


Fig. 8.7(a). $v(x = 1.0, y)$ for $Re = 100$.

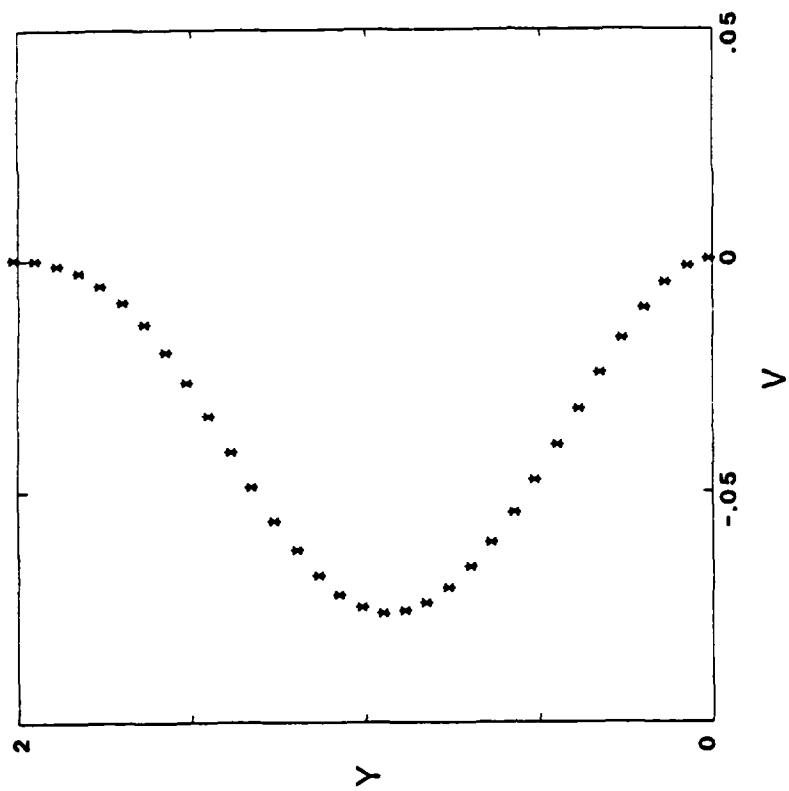


Fig. 8.7(b). $v(x = 4.0, y)$ for $Re = 100$.

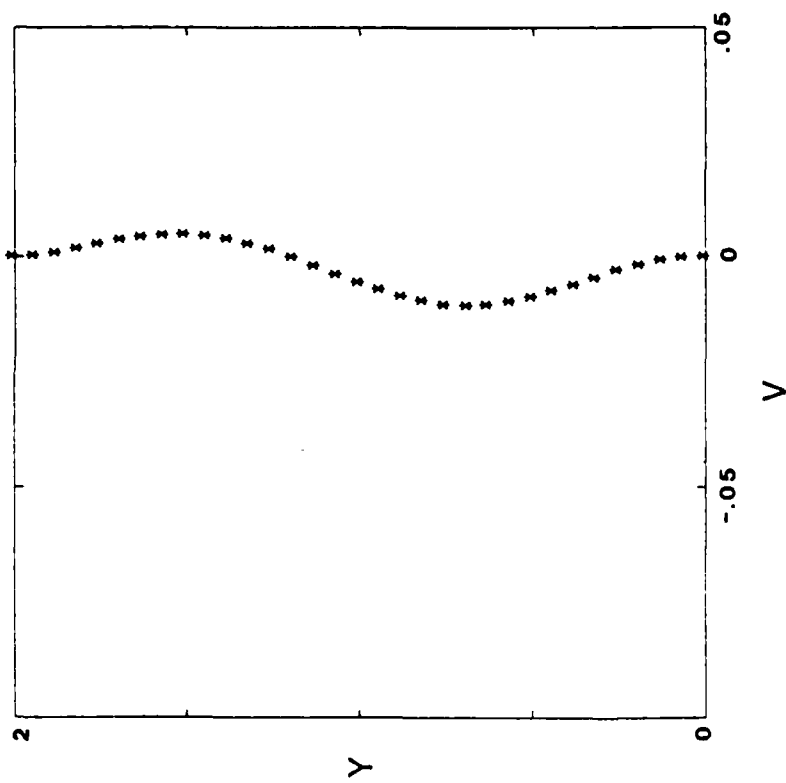


Fig. 8.7(c). $v(x = 6.0, y)$ for $Re = 100$.

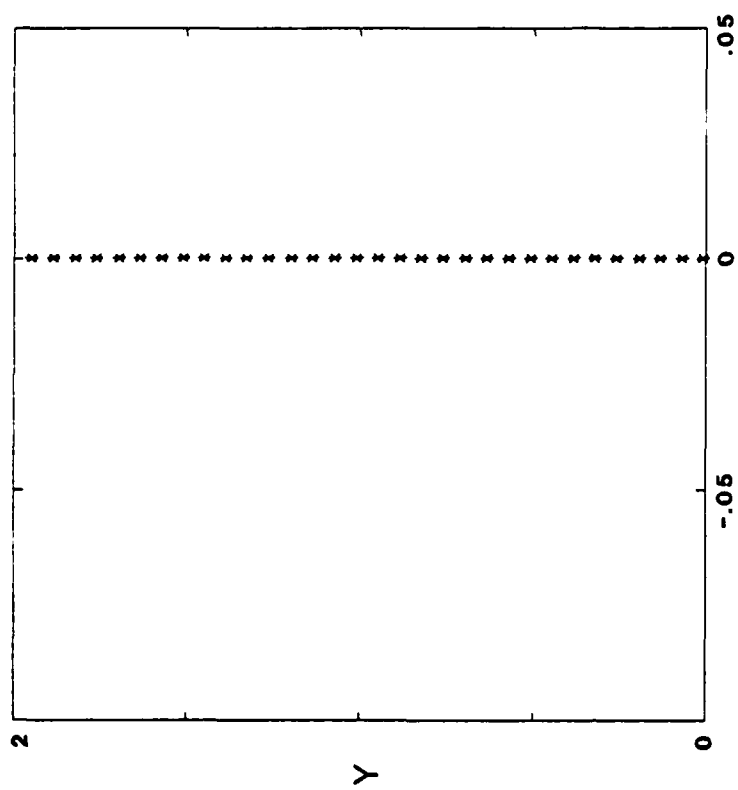


Fig. 8.7(d). $v(x = 12.0, y)$ for $Re = 100$.

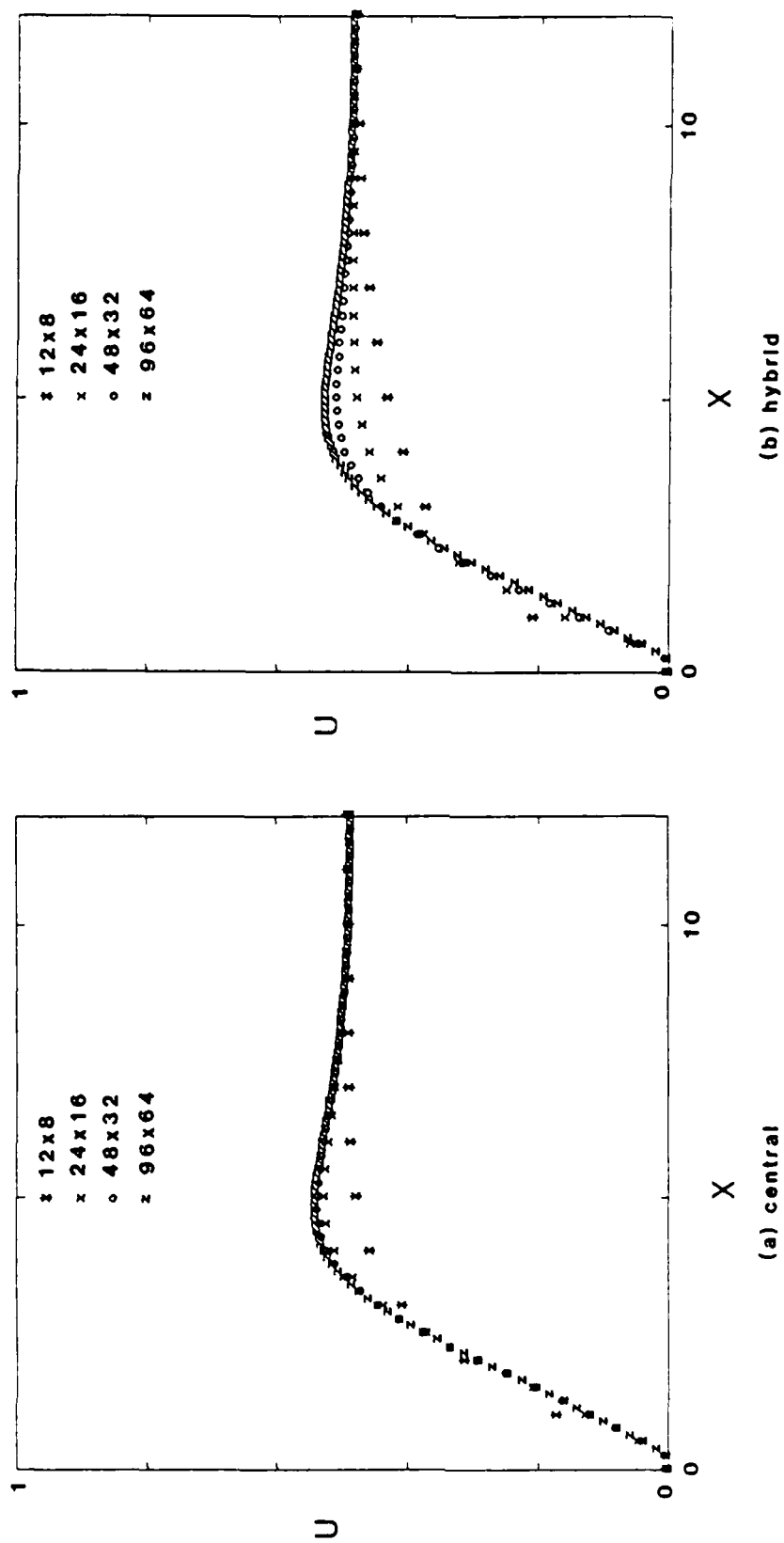


Fig. 8.8. Comparison of difference schemes; $u(x, \Delta x, y = 0.8125)$.

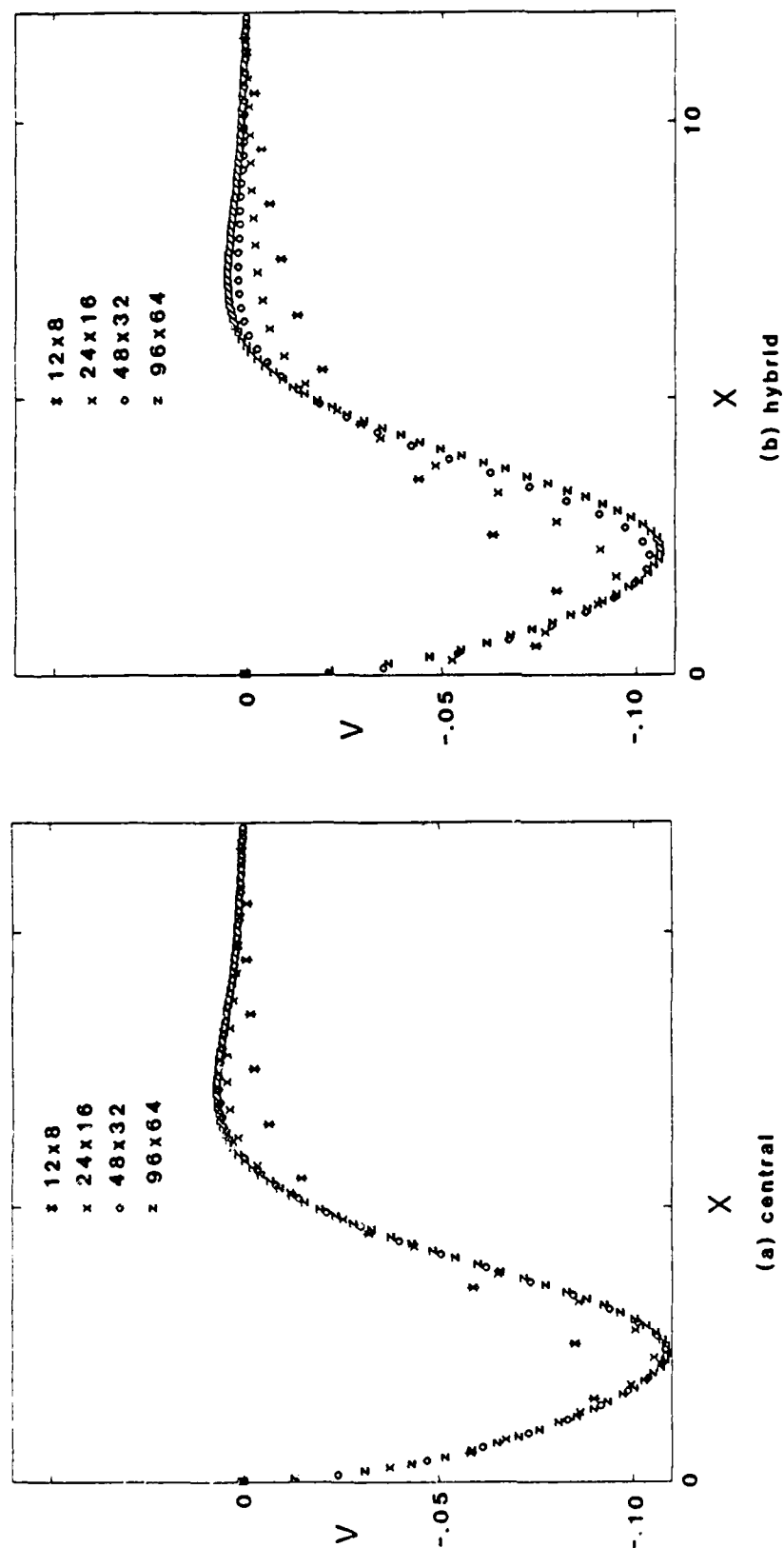
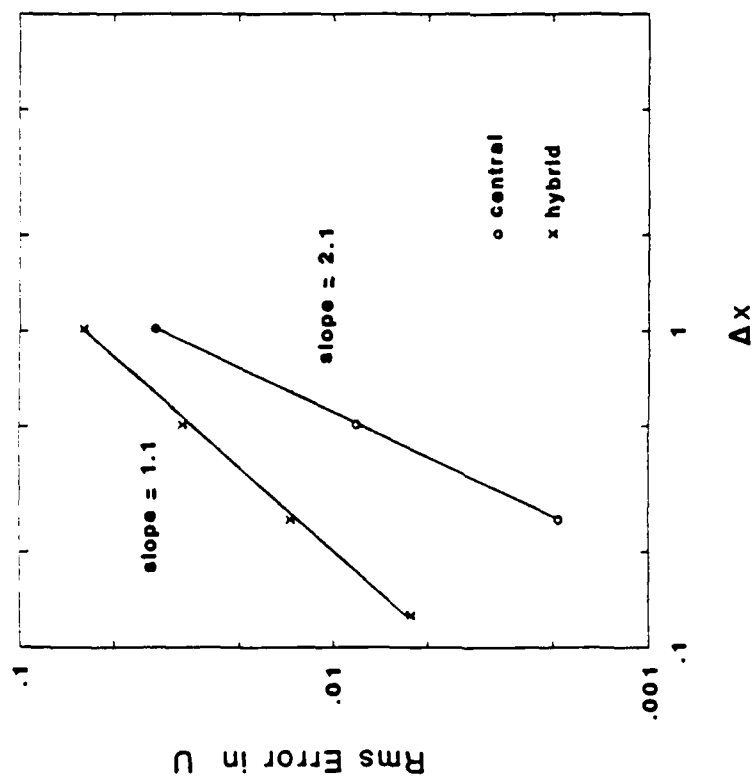
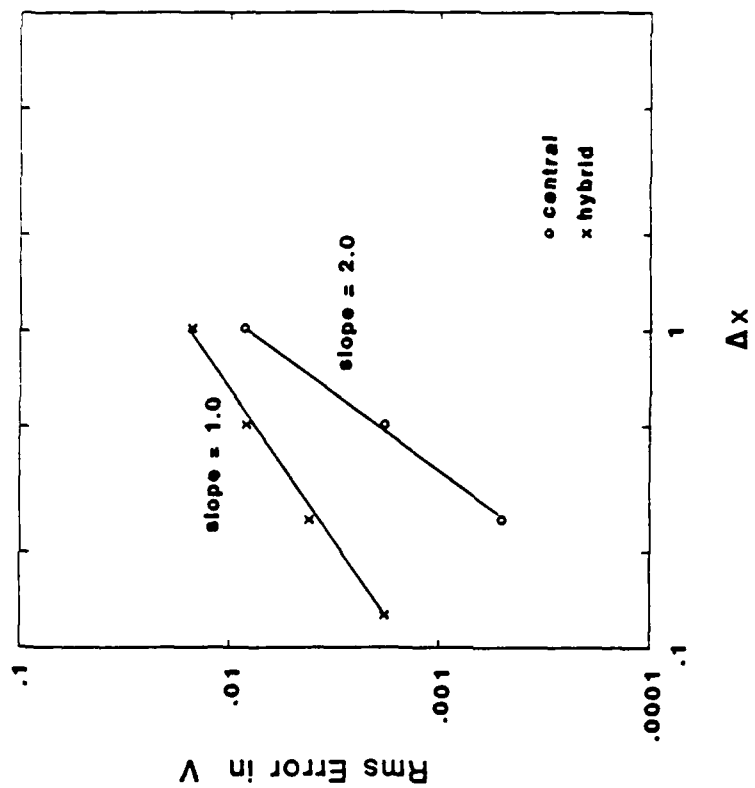


Fig. 8.9. Comparison of difference schemes; $v(x, \Delta x, y = 1.25)$.

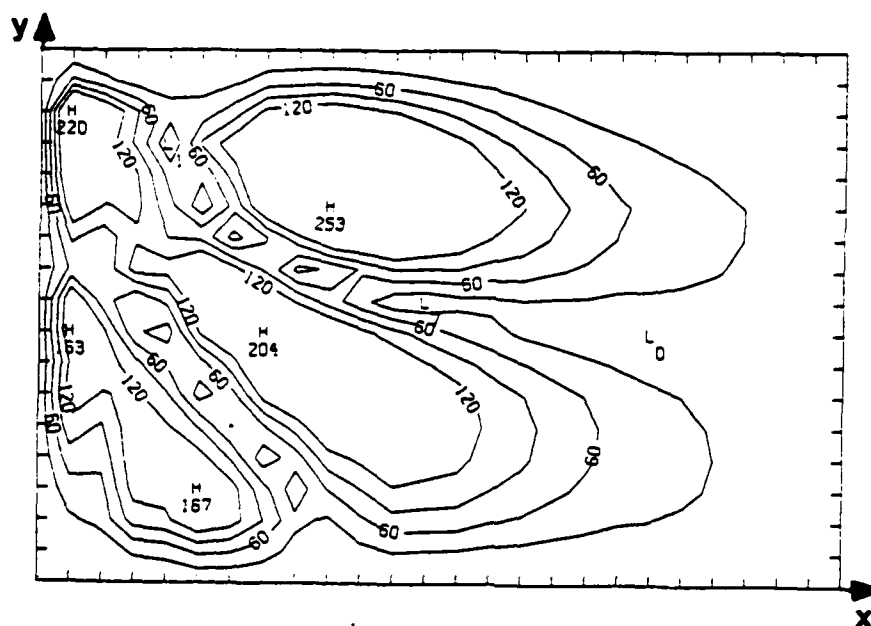


(a)

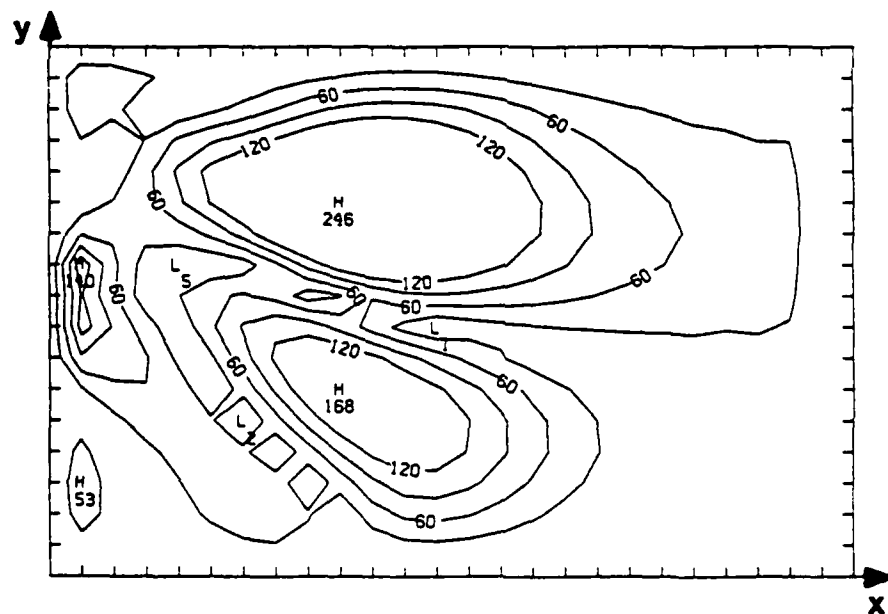


(b)

Fig. 8.10. Comparison of difference schemes; error vs. Δx .



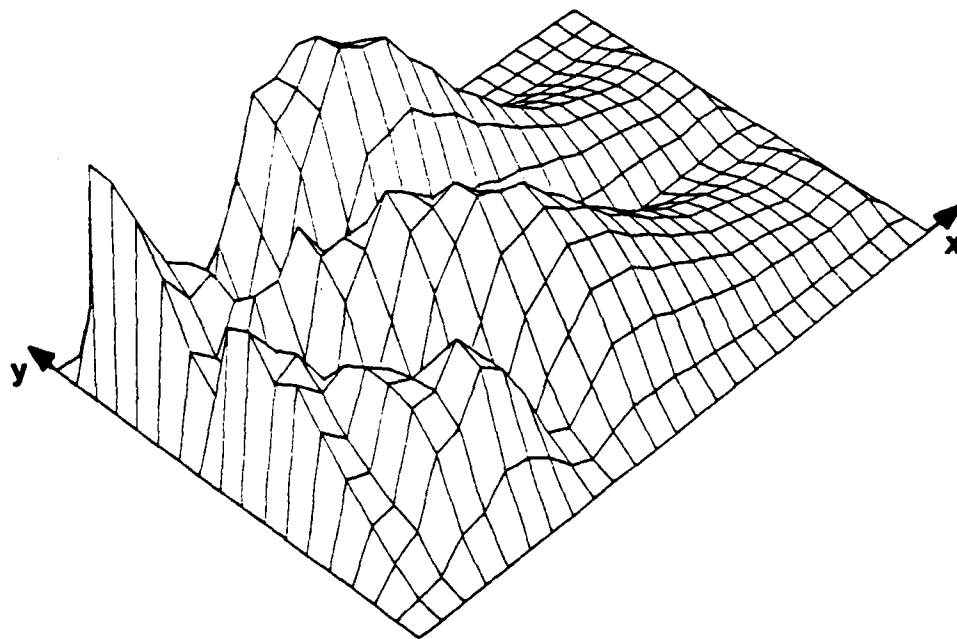
estimated



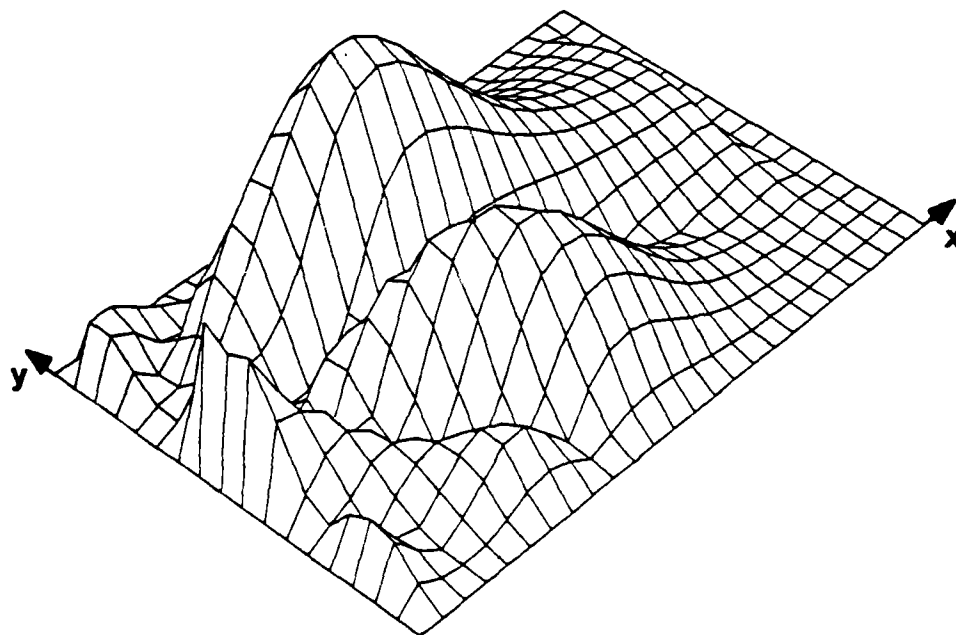
exact

(a) contour view

Fig. 8.11. Solution error in u for $Re = 100$.



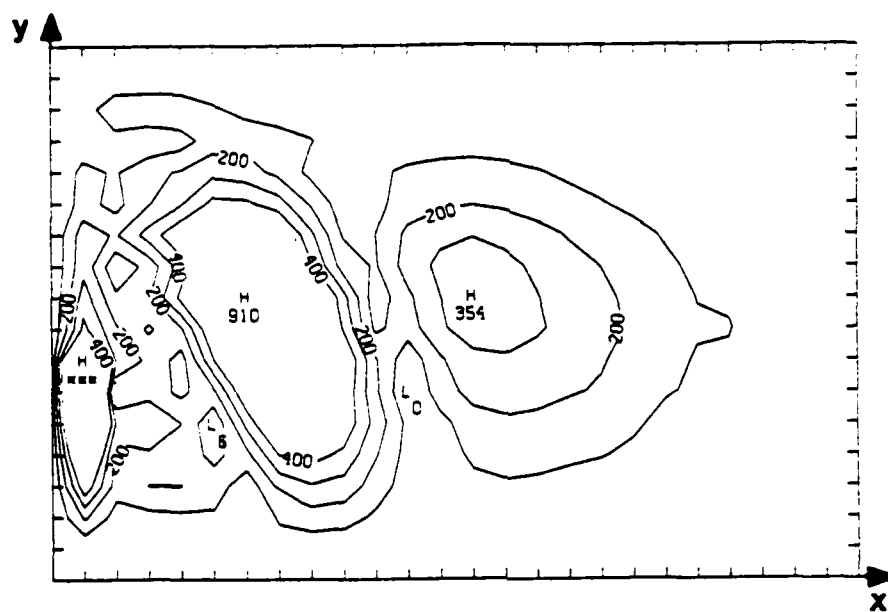
estimated



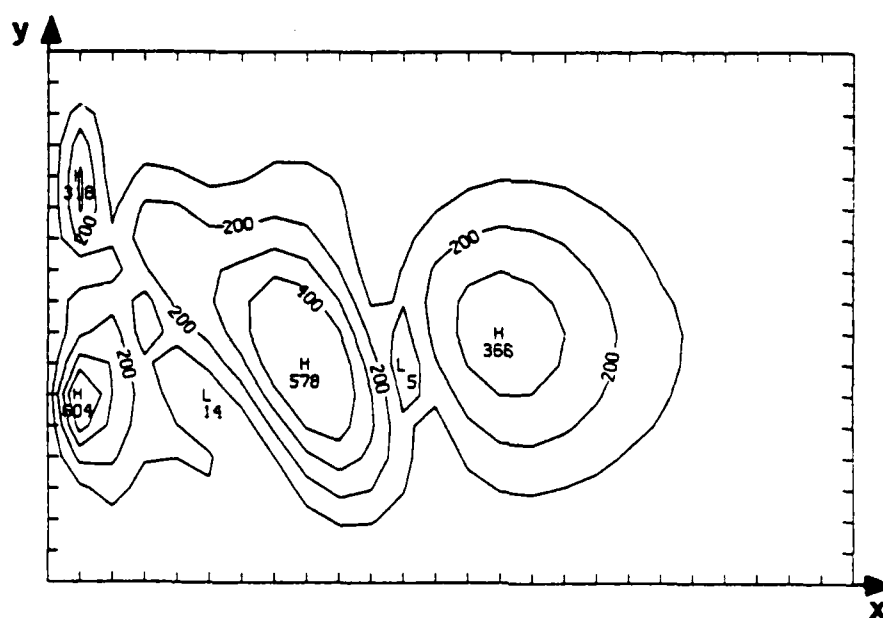
exact

(b) surface view

Fig. 8.11. Solution error in u for $Re = 100$.



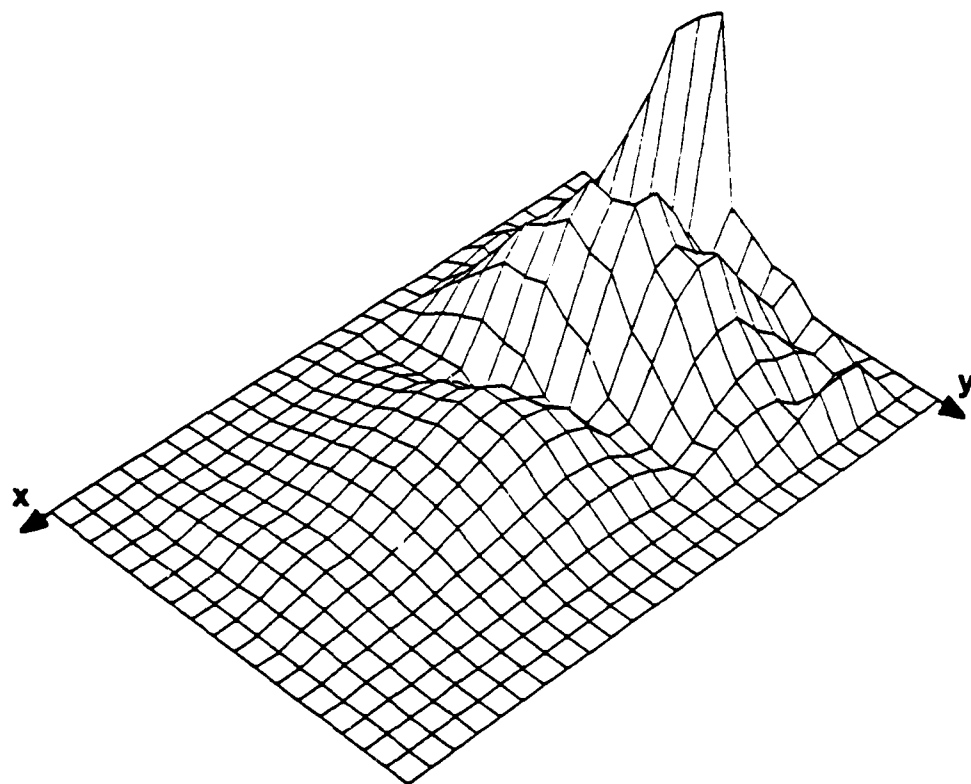
estimated



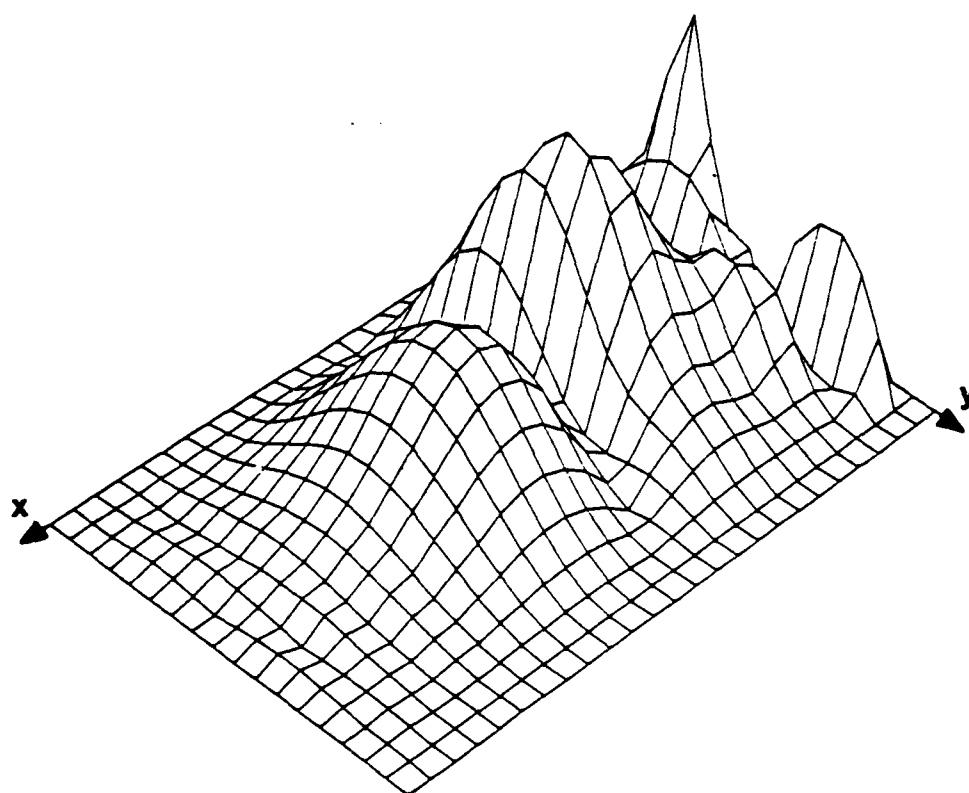
exact

(a) contour view

Fig. 8.12. Solution error in v for $Re = 100$.



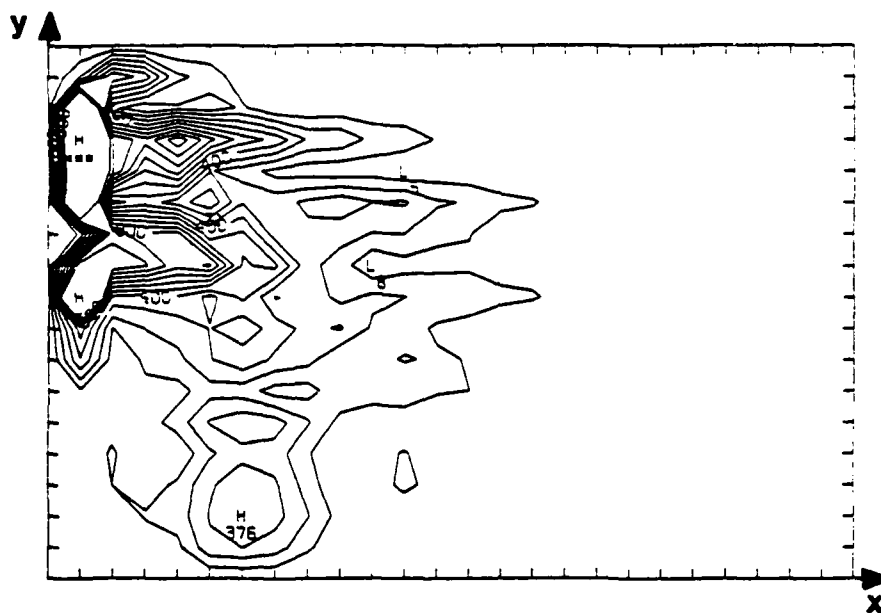
estimated



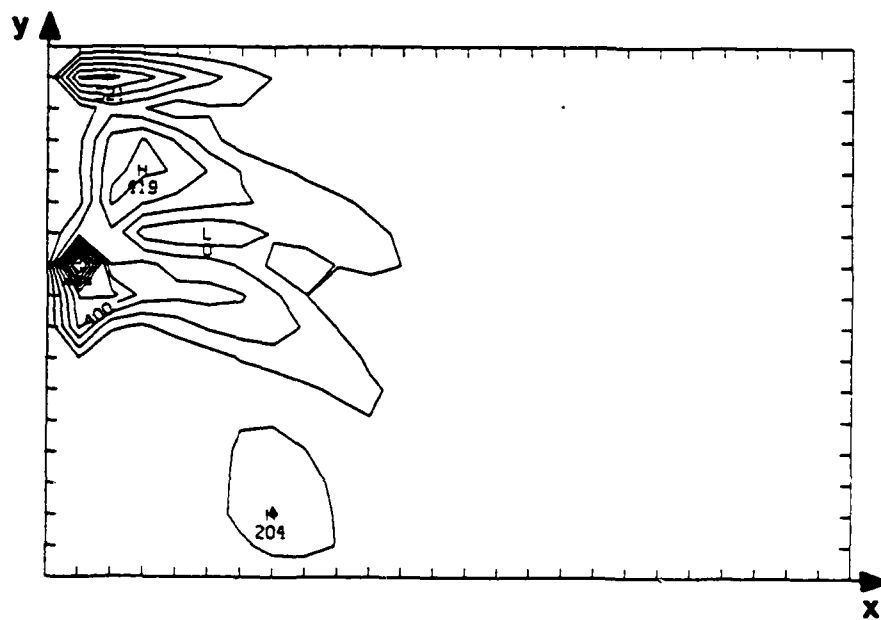
exact

(b) surface view

Fig. 8.12. Solution error in v for $Re = 100$.



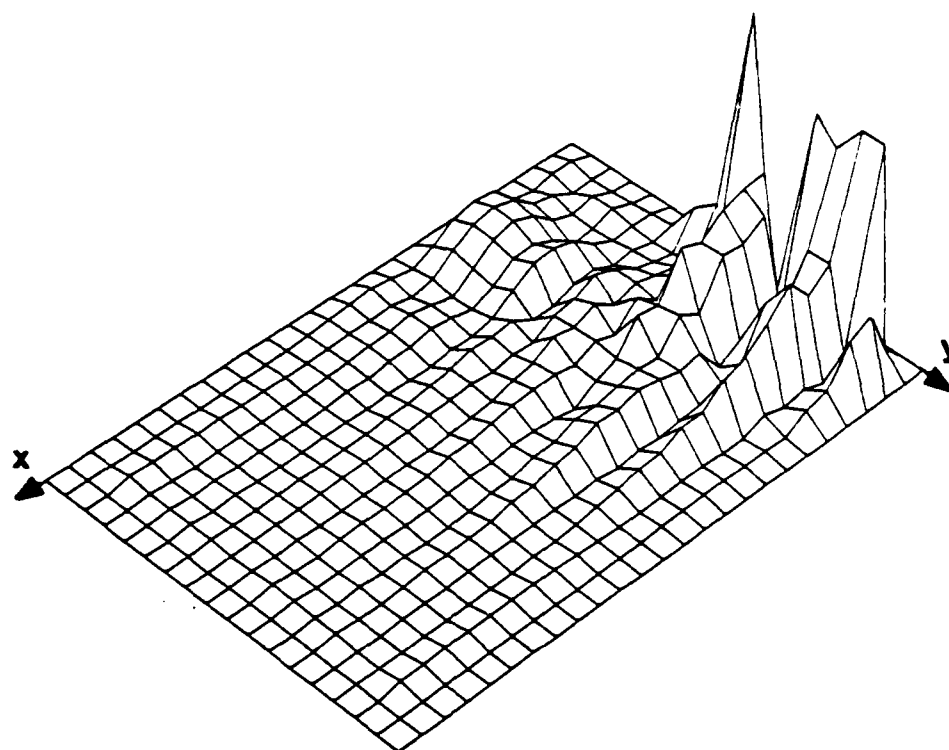
estimated



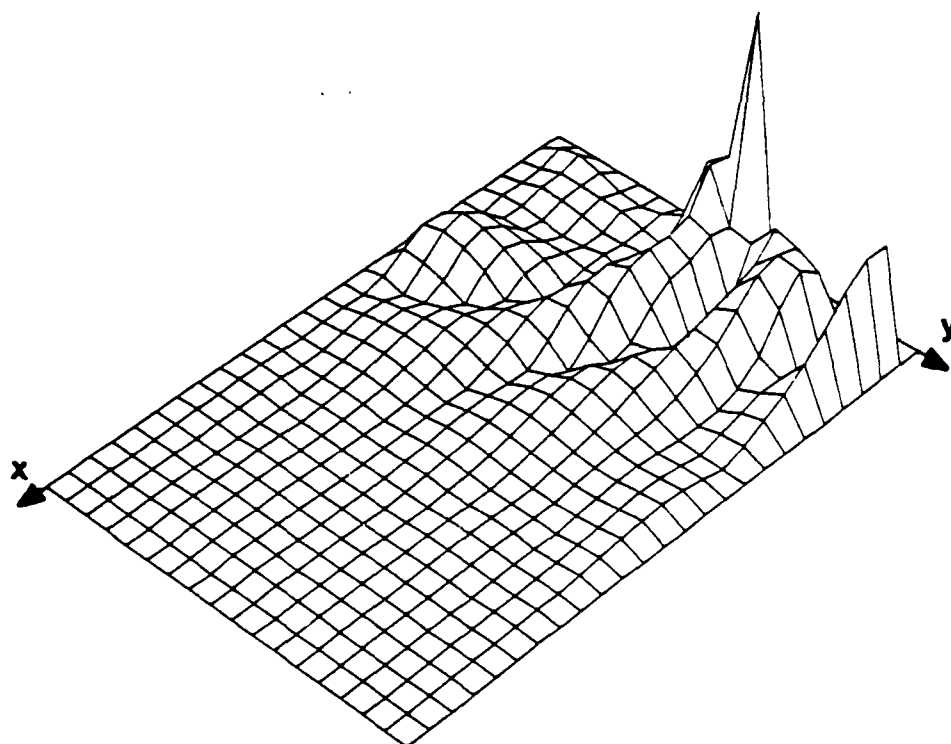
exact

(a) contour view

Fig. 8.13. Truncation error in u for $Re = 100$.



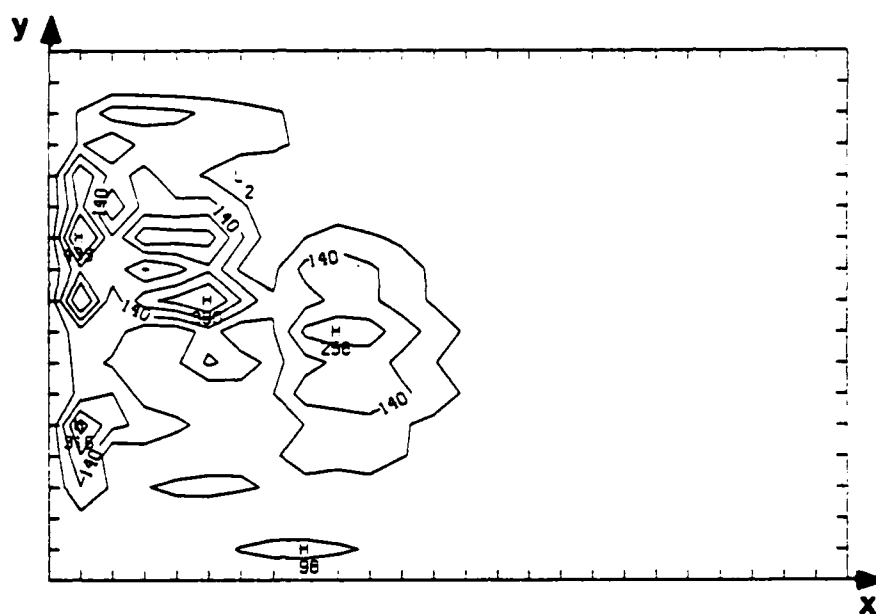
estimated



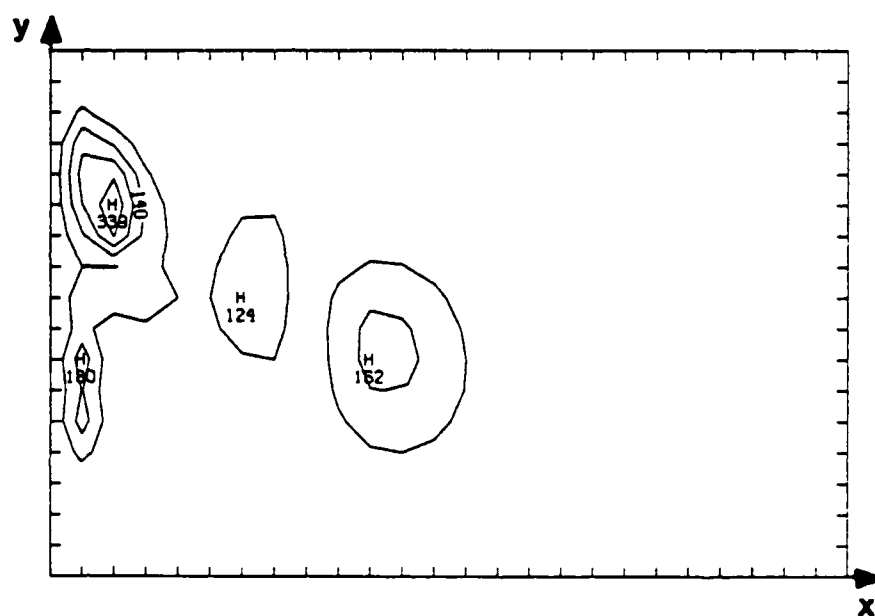
exact

(b) surface view

Fig. 8.13. Truncation error in u for $Re = 100$.



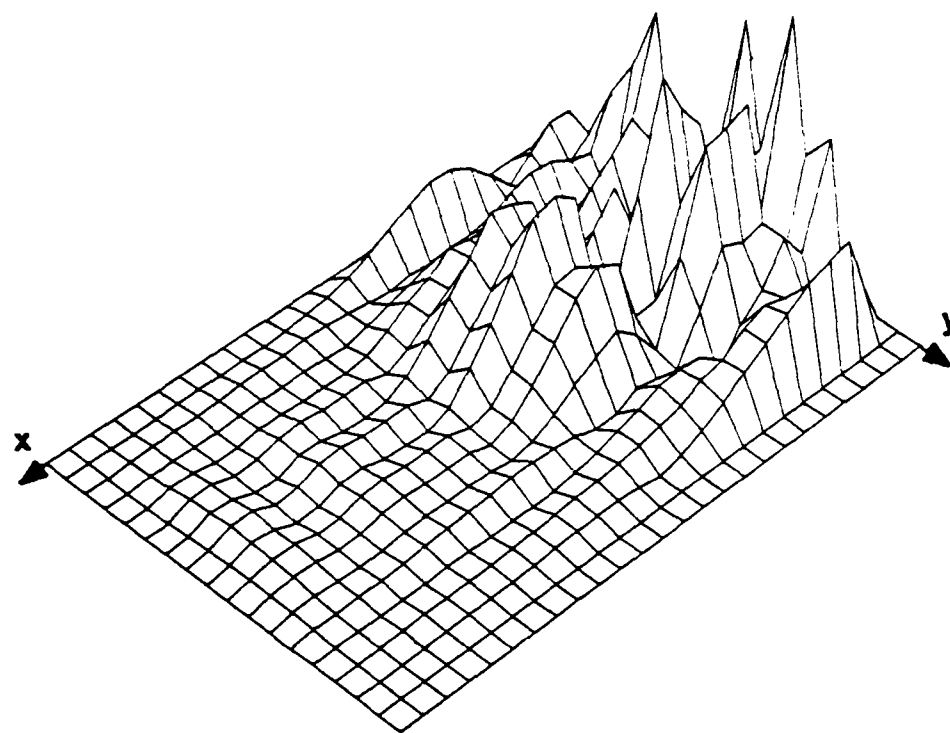
estimated



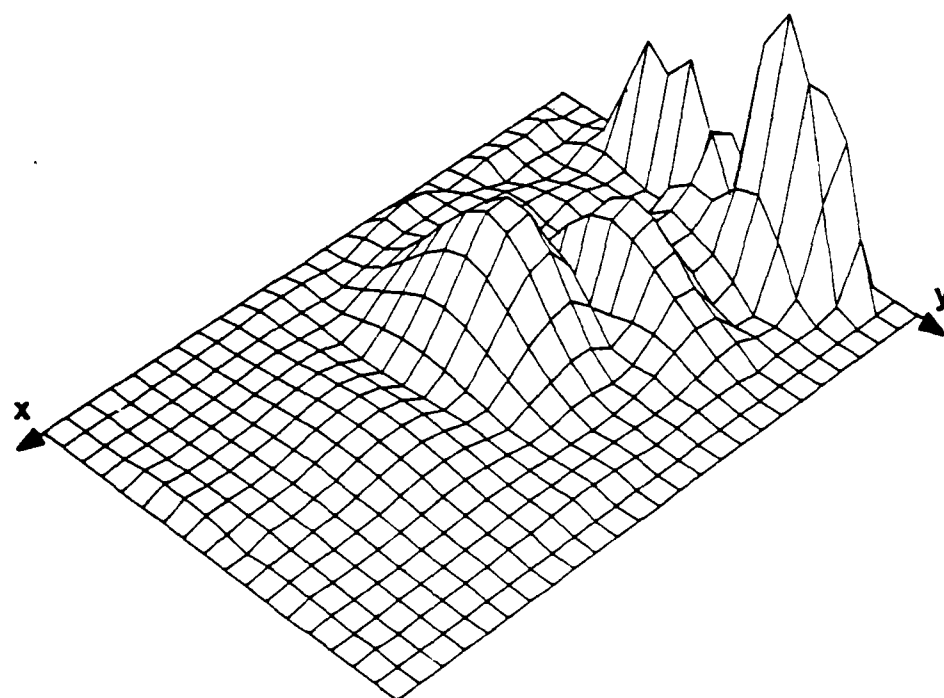
exact

(a) contour view

Fig. 8.14. Truncation error in v for $Re = 100$.



estimated



exact

(b) surface view

Fig. 8.14. Truncation error in v for $Re = 100$.

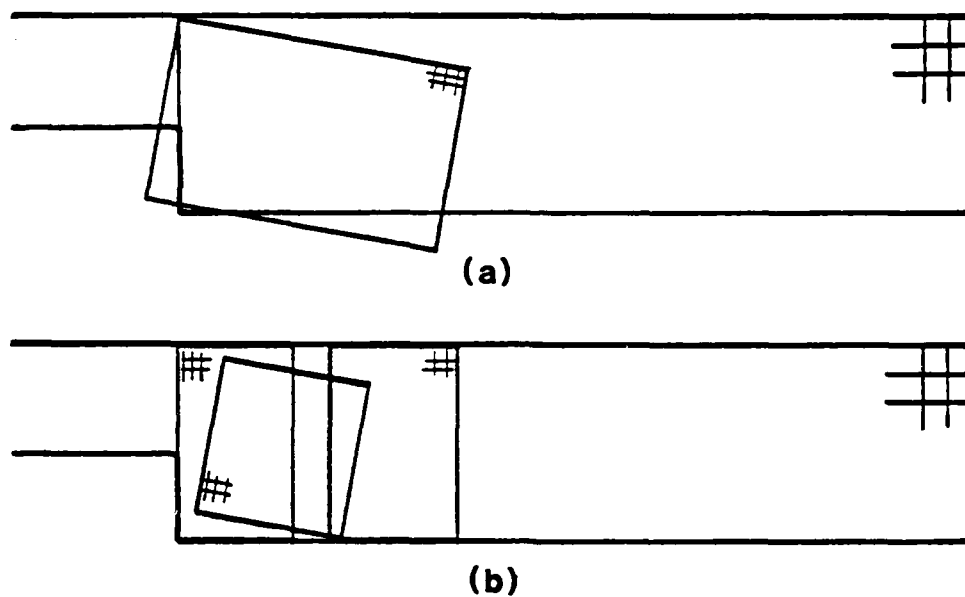


Fig. 8.15. Rotated refined rectangles for backstep.

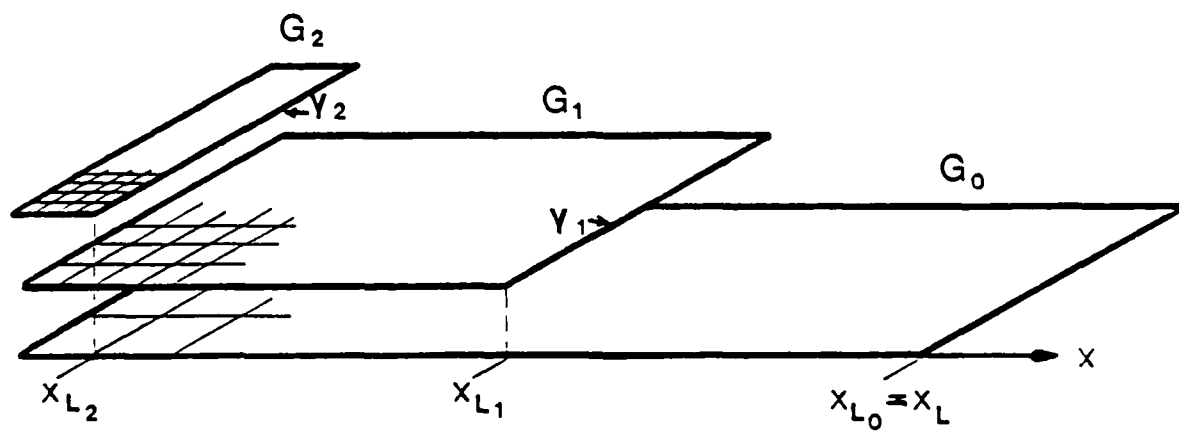


Fig. 8.16. Notation for refined grids.

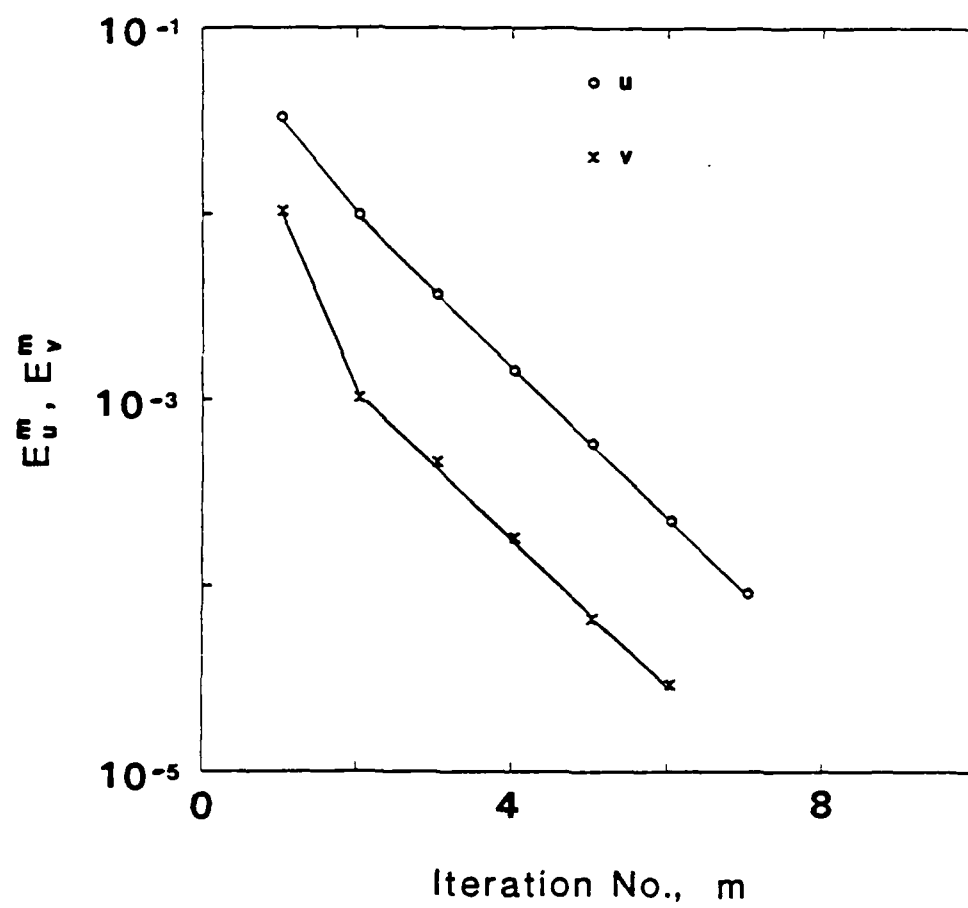
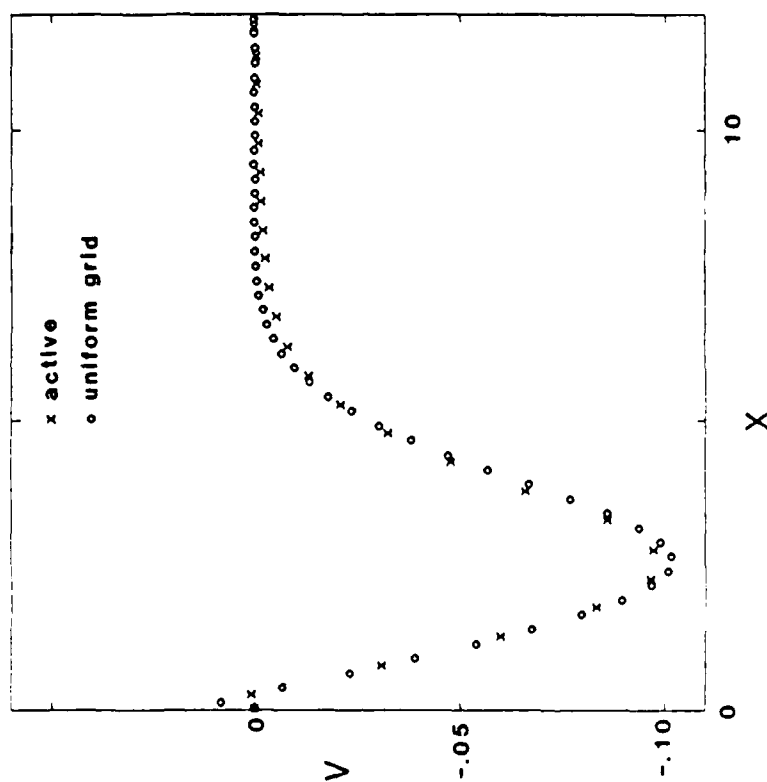
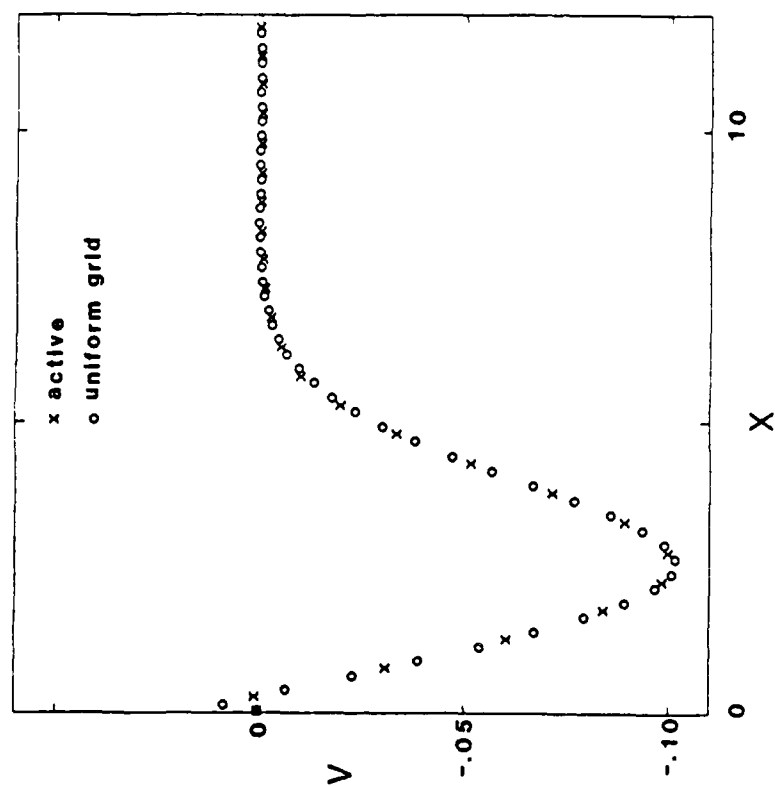


Fig. 8.17. Convergence of active method on two-level grid system.



(a) $x_{L,1} = 4$



(b) $x_{L,1} = 6$

Fig. 8.18 Effect of internal boundary location

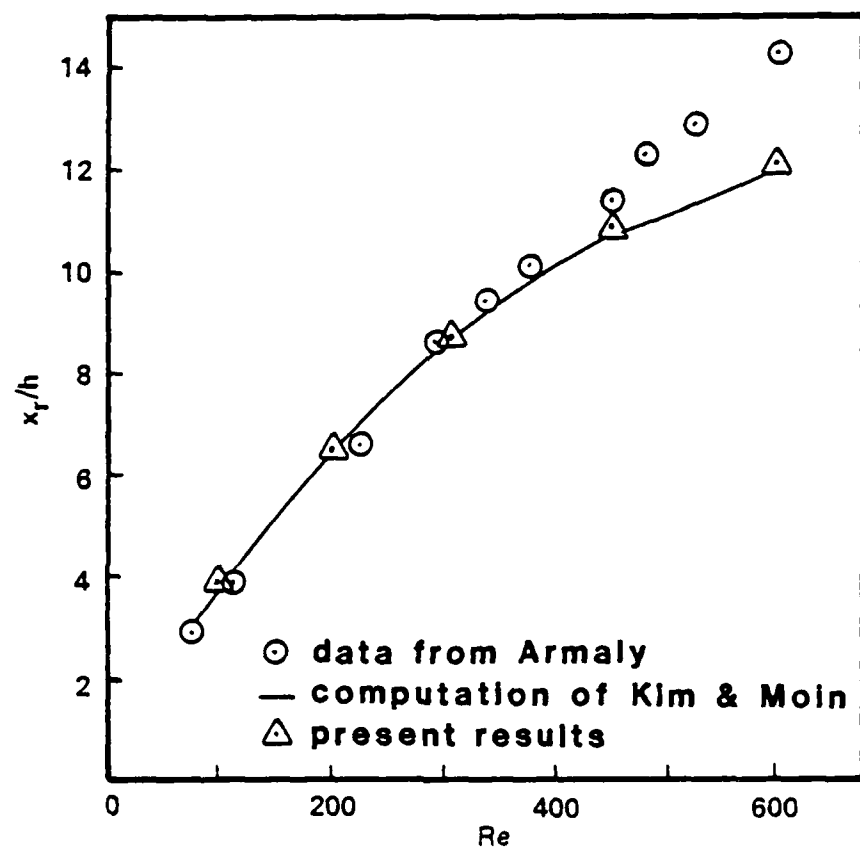


Fig. 8.19. Reattachment length vs. Reynolds number.

AD-A169 471

ADAPTIVE GRID TECHNIQUES FOR ELLIPTIC FLUID-FLOW
PROBLEMS(U) STANFORD UNIV CA CENTER FOR LARGE SCALE
SCIENTIFIC COMPUTATION S C CARUSO ET AL. DEC 85
CLASSIC-85-10 N00014-82-K-0335

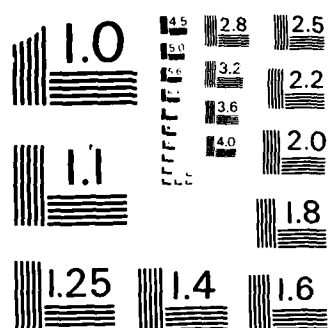
3/3

UNCLASSIFIED

F/G 20/4

NL





MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

CONCLUSIONS AND RECOMMENDATIONS

In this research, we have developed adaptive grid techniques for flows governed by the incompressible Navier-Stokes equations. The local refinement method of Berger, developed for hyperbolic equations, was extended to elliptic equations. Specifically, we implemented two features of Berger's method: overlaid, locally uniform grid refinement consisting of rotated rectangles, and refinement regions defined by Richardson error estimates.

Local refinement offers the following important advantages over global refinement methods:

- lower computational overhead,
- no instability or skewness problems associated with grid-point distribution,
- truncation error used for refinement criterion,
- less reliance on heuristic criteria and problem-dependent parameters,
- solution accuracy explicitly addressed.

Two classes of elliptic flows were identified; they are characterized as having strong or weak viscous-inviscid interactions. Adaptive solution methods, active and passive, respectively, were developed for each class.

Application of the passive method to linear, two-point boundary-value problems demonstrated the feasibility of the solution technique. The method is efficient compared to using a uniform fine grid.

The passive method was also applied to a 2-D, linear convection-diffusion problem, in which the flow is oblique to the grid lines. The refined grids automatically aligned with the flow, thereby minimizing numerical diffusion. For fixed accuracy, the adaptive method is significantly more efficient than using a uniform grid.

The SIMPLER method was used to solve the steady, laminar, incompressible Navier-Stokes equations. Central differencing of the convective terms was implemented with the defect-correction method to

stabilize the solution method for all cell Reynolds numbers. Smooth solutions were calculated for cell Reynolds numbers as high as 150, indicating that the commonly used restriction, $Re_{\Delta x} < 2$, is too severe.

Uniform grid calculations were performed for the laminar backstep flow. Patankar's power-law scheme was shown to be less accurate than central differencing, and only first-order accurate.

The Richardson-estimated solution and truncation errors were also compared to accurate estimates of the same quantities for the backstep flow. The solution error is predicted well. The truncation-error estimates are less accurate, but they reliably indicate where grid refinement is required.

Active-adaptive calculations of the backstep were made, using boundary-aligned refinement. At $Re = 100$, the adaptive calculation has comparable accuracy but is six times faster than a uniform-grid calculation. Adaptive calculations were also made at higher Reynolds numbers. The calculations agree well with the experimental data and other calculations.

Recommendations

Several aspects of our adaptive technique should be further studied and improved. First, the method has been developed for rectangular problem domains; extension to complex geometries will increase the method's usefulness as an engineering tool.

Interpolation methods for rotated grids need to be developed. The issue of conservation and how it is maintained using local grid refinement needs to be better understood. Also, to increase the adaptive efficiency, higher-order interpolation methods, preferably cubic interpolation, for fine-grid boundary conditions are recommended.

The convergence rate of the active solution method is probably not optimum. It should be determined what is the best iteration strategy--to iterate for a fixed number of times (and how many) or to iterate to a partial convergence on each grid before switching. The choice probably depends on the problem and also the Reynolds number.

Since the active method is similar to multigrid methods (which are very efficient), the techniques used to improve multigrid convergence rates may be applicable to the active method; this should be further evaluated. For example, different sweep patterns may prove more efficient; e.g., "W" sweeps, the so-called Full Multigrid (FMG) cycle, etc. Also, when switching from a coarse to a fine grid in a multigrid method, the fine grid solution is corrected at all internal fine grid points. Recall that, in the active method, only the fine grid boundary conditions are interpolated from the coarse grid. Addition of such a correction may improve the overall convergence rate.

Finally, the method should be extended to the Reynolds-averaged equations and also to three dimensions. Such flows pose some of the most challenging engineering problems.

REFERENCES

- Allen, D. N. de G., and R. V. Southwell (1955), "Relaxation Methods Applied to Determine the Motion, in Two Dimensions, of a Viscous Fluid Past a Fixed Cylinder," Q. J. Mech. Appl. Math., Vol. 8, p. 129.
- Anderson, D. A. (1983). "Adaptive Grid Methods for Partial Differential Equations," in Advances in Grid Generation, ASME Fluids Engrg. Conference, Houston.
- Anderson, D. A., and M. M. Rai (1982), "The Use of Solution Adaptive Grids in Solving Partial Differential Equations," in Numerical Grid Generation (J. F. Thompson, ed.), North-Holland, New York.
- Armaly, B. F., F. Durst, J. C. F. Pereira and B. Schonung (1983), "Experimental and Theoretical Investigation of Backward-Facing Step Flow," Jnl. Fluid Mech., Vol. 127, pp.473-496.
- Atta, E. H., and J. Vadyak (1983), "A Grid Overlapping Scheme for Flow-field Computations about Multicomponent Configurations," AIAA Jnl., Vol. 21, No.1, pp.1271-1277.
- Auzinger, W., and H. J. Stetter (1982), "Defect Corrections and Multigrid Iterations," in Lecture Notes in Mathematics, #960: Multigrid Methods, Springer-Verlag.
- Babuska, I., J. Chandra and J. E. Flaherty (1983), Adaptive Computational Methods for Partial Differential Equations, SIAM, Philadelphia.
- Bai, D. (1984), "Local Mesh Refinement Multilevel Techniques," Ph.D. Thesis, Dept. of Applied Mathematics, Weizmann Institute, Israel.
- Bender, C. M. and S. A. Orszag (1978), Advanced Mathematical Methods for Scientists and Engineers, McGraw-Hill, New York.
- Berger, M. J. (1982), "Adaptive Mesh Refinement for Hyperbolic Partial Differential Equations," Ph.D. Thesis, Dept. of Computer Science, Stanford University, Calif.
- Berger, M. J. (1983), "Data Structures for Adaptive Mesh Refinement," in Adaptive Computational Methods for Partial Differential Equations (I. Babuska, J. Chandra and J. E. Flaherty, eds.), SIAM, Philadelphia.
- Berger, M. J. (1984a), personal communication.
- Berger, M. J. (1984b), "On Conservation at Grid Interfaces," ICASE Rept. No. 84-83, NASA Langely Res. Ctr.
- Berger, M. J. and J. Oliger (1984), "Adaptive Mesh Refinement for Hyperbolic Partial Differential Equations," Jnl. Comp. Phy., Vol. 53, pp.484-512.

- Berger, M. J. and A. Jameson (1985), "Automatic Adaptive Grid Refinement for the Euler Equations," AIAA Jnl., Vol. 23, No.4, pp. 561-568.
- Bolstad, J. H. (1982), "An Adaptive Finite Difference Method for Hyperbolic Systems in One Space Dimension," Ph.D. Thesis, Dept. of Computer Science, Stanford University, Calif.
- Brackbill, J. U. (1982), "Coordinate System Control: Adaptive Meshes," in Numerical Grid Generation (J. F. Thompson, ed.), North-Holland, New York.
- Brandt, A. (1977), "Multi-Level Adaptive Solutions to Boundary Value Problems," Math. Comp., Vol. 31, No. 138, pp. 333-390.
- Brandt, A. (1982), "Guide to Multigrid Development," in Lecture Notes in Mathematics, #960: Multigrid Methods, Springer-Verlag.
- Brown, D. L. (1982), "Solution Adaptive Mesh Procedures for the Numerical Solution of Singular Perturbation Problems," Ph.D. Thesis, Dept. of Applied Mathematics, California Institute of Technology, Calif.
- Chorin, A. J. (1968), "Numerical Solution of the Navier-Stokes Equations," Math. Comp., Vol. 22, pp. 745-762.
- Dihn, Q. V., R. Glowinski, and J. Periaux (1984), "Solving Elliptic Problems by Domain Decomposition Methods with Applications," in Elliptic Problem Solvers II (Birkhoff and A. Schoenstadt, eds.), Academic Press, Orlando, Fla.
- Dorr, F. W. (1970), "The Numerical Solution of Singular Perturbations of Boundary Value Problems," SIAM Jnl. Numer. Anal., Vol.7, No.2, June.
- Durst, F. and J.C.F. Pereira (1983), "Calculation of Laminar Backward-Facing Step Flow with Four Discretization Schemes for the Convection Terms," GAMM Workshop, Bievres, France.
- Dwyer, H. A. (1984), "Grid Adaption for Problems in Fluid Dynamics," AIAA Jnl., Vol. 22, No. 12, pp. 1705-1712.
- Dwyer, H. A., R. J. Kee, and B. R. Sanders (1980), "Adaptive Grid Methods for Problems in Fluid Mechanics and Heat Transfer," AIAA Jnl., Vol. 18, No. 10, pp. 1205-1212.
- Dwyer, H. A., M. D. Smooke, and R. J. Kee (1982), "Adaptive Gridding for Finite Difference Solutions to Heat and Mass Transfer Problems," in Numerical Grid Generation (J.F. Thompson, ed.), North-Holland, New York.
- Ferziger, J. H. (1983), "Higher Level Simulations of Turbulent Flow," in Computational Methods for Turbulent, Transonic and Viscous Flows, (J.-A. Eessers, ed.), Hemisphere Publishing Corp, New York.

- Ferziger, J. H. (1986), "Simulation of Incompressible Turbulent Flows," paper submitted to Jnl. Comp. Phy.
- Fuchs, L. (1985), "Multi-Grid Solutions on Grids with Non-Aligning Coordinates," paper presented at Second Copper Mountain Conference on Multigrid Methods, April 1-3, Copper Mountain, Colorado.
- Gladwell, I., and R. Wait (1979), A Survey of Numerical Methods for Partial Differential Equations, Clarendon Press, Oxford.
- Gnoffo, P. A. (1982), "A Vectorized, Finite Volume, Adaptive-Grid Algorithm for Navier-Stokes Calculations," in Numerical Grid Generation, (J. F. Thompson, ed.) North-Holland, New York.
- Gnoffo, P. A. (1983), "A Finite Volume, Adaptive Grid Algorithm Applied to Planetary Entry Flowfields," AIAA Jnl., Vol. 21, No. 9, pp. 1249-1254.
- Greenburg, J. B. (1985), "A New Self-Adaptive Grid Method," AIAA Jnl., Vol. 23, No. 2, pp. 317-320.
- Gropp, W. D. (1980), "A Test of Moving Mesh Refinement for 2-D Scalar Hyperbolic Problems," SIAM Jnl. Sci. Stat., Vol. 1, No. 2, pp. 191-197.
- Hageman, L. A., and D. M. Young (1981), Applied Iterative Analysis, Academic Press, New York.
- Han, T., J.A.C. Humphrey, and B. E. Launder (1981), "A Comparison of Hybrid and Quadratic-Upstream Differencing in High Reynolds Number Elliptic Flows," Comp. Meth. Appl. Mech. and Eng., Vol. 29, pp. 81-95.
- Harlow, F. T., and J. E. Welch, "Numerical Calculations of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface," Phy. Fluids, Vol. 8, No. 12, pp. 2182-2189.
- Hedstrom, G. W., and G. H. Rodrigue (1982), "Adaptive Methods for Time-Dependent Partial Differential Equations," in Lecture Notes in Mathematics #960: Multigrid Methods, Springer-Verlag.
- Hemker, P. W. (1981), "An Accurate Method Without Directional Bias for the Numerical Solution of a 2-D Elliptic Singular Perturbation Problem," Rept. NW 117/81, Dept. of Numerical Mathematics, Center of Mathematics, Amsterdam.
- Hessenius, K. A., and M. M. Rai (1984), "Applications of a Conservative Zonal Scheme to Transient and Geometrically Complex Problems," AIAA Paper No. 84-1532.
- Il'in, A. M. (1969), "Differencing Scheme for a Differential Equation with a Small Parameter Affecting the Highest Derivative," Math. Notes, Vol. 6, pp. 596-602.

- Jameson, A., "Iterative Solution of Transonic Flows over Airfoils and Wings, Including Flows at Mach 1," Comm. Pure Appl. Math., Vol. XXVII, pp. 283-309.
- Kang, L. S., Y. P. Chen, L. L. Sun, and H. Y. Quan (1985), "A Class of New Asynchronous Parallel Algorithms for Solving Partial Differential Equations," Mathematical Research Report No. 19, Wuhan University, Wuhan, China, (in English).
- Kellog, R. B., and A. Tsan (1978), "Analysis of Some Difference Approximations for a Singular Perturbation Problem Without Turning Points," Math. Comp., Vol. 32, No. 144, pp. 1025-1039.
- Kim, J., and P. Moin (1985), "Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations," Jnl. Comp. Phys., Vol. 59, pp. 308-323.
- Kline, S. J., B. J. Cantwell, and G. M. Lilley (1982), The 1980-81 AFOSR-HTTM- Stanford Conference on Complex Turbulent Flows, Mechanical Engrg. Dept., Stanford University, Stanford, CA.
- Kreiss, B., and H. O. Kreiss (1981), "Numerical Methods for Singular Perturbation Problems," SIAM Jnl. Numer. Anal., Vol. 18, No.2, pp. 262-276.
- Leonard, B. P. (1979a), "A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation," Comp. Meth. Appl. Mech. and Eng., Vol. 19, pp. 59-98.
- Leonard, B. P. (1979b), "A Survey of Finite Differences of Opinion on Numerical Muddling of the Incomprehensible Defective Confusion Equation," in Finite Element Methods in Convection Dominated Flows, (T. Hughes, ed.), ASME WAM, New York.
- Leschziner, M. A. (1980), "Practical Evaluation of Three Finite Difference Schemes for the Computation of Steady-state Recirculating Flows," Comp. Meth. Appl. Mech. and Eng., Vol. 23, pp. 293-312.
- Leschziner, M. A., and W. Rodi (1981), "Calculation of Annular and Twin Parallel Jets Using Various Discretization Schemes and Turbulence-Model Variations," Jnl. Fluids Eng. Vol. 103, pp. 352-360.
- Mastin, C. W. (1982), "Error Induced by Coordinate Systems," in Numerical Grid Generation (J.F. Thompson, ed.), North-Holland, New York.
- Mehta, U. B. (1984), "Physical Aspects of Computing the Flow of a Viscous Fluid," NASA TM 85893, Ames Res. Ctr.
- Miller, K. (1965), "Numerical Analogs to the Schwarz Alternating Procedure," Numer. Math., Vol. 7, pp. 91-103.
- Murman, E. M., and J. R. Baron (1983), "Computational Methods for Complex Flowfields," 1983 Annual Report, AFOSR-TR-83-0841

- Nakahashi, K. and G. S. Diewert (1984), "A Practical Adaptive-Grid Method for Complex Fluid-Flow Problems," NASA TM-85989, Ames Res. Ctr.
- Nakahashi, K., and G. S. Diewert (1985), "A Three-Dimensional Adaptive Grid Method," AIAA Paper No. 85-0486.
- Olinger, J. (1984), "Adaptive Grid Methods for Hyperbolic Partial Differential Equations," in Inverse Problems of Acoustic and Elastic Waves (F. Santosa, Y.-H. Pao, W. W. Symes and C. Holland, eds.), SIAM, Philadelphia.
- Patankar, S. V. (1980), Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp., New York.
- Patankar, S. V. (1981), "A Calculation Procedure for Two-Dimensional Elliptic Situations," Numerical Heat Transfer, Vol. 4, pp.409-425.
- Pearson, C. E. (1968), "On a Differential Equation of Boundary Layer Type," Jnl. Math. and Phys., Vol. 47, pp. 134-154.
- Pearson, C. E. (1981), "Use of Streamline Coordinates in the Numerical Solution of Compressible Flow Problems," Jnl. Comp. Phys., Vol. 42, pp. 257-265.
- Peyret, R., and T. D. Taylor (1983), Computational Methods for Fluid Flow, Springer-Verlag, New York.
- Pierson, B. L., and P. Kutler (1980), "Optimal Nodal Point Distribution for Improved Accuracy in Computational Fluid Dynamics," AIAA Jnl., Vol 18, No. 1, pp. 49-54.
- Rai, M. M (1985), "An Implicit, Conservative, Zonal-Boundary Scheme for Euler Equation Calculations," AIAA Paper No. 85-0488.
- Rai, M. M., and D. A. Anderson (1981a), "Grid Evolution in Time Asymptotic Problems," Jnl. Comp. Phys., Vol. 43, pp. 327-344.
- Rai, M. M., and D. A. Anderson (1981b), "The Use of Adaptive Grids in Conjunction with Shock Capturing Methods," AIAA Computational Fluid Dynamics Conference, Palo Alto, CA.
- Raithby, G. D. (1976), "Skew-upstream Differencing Schemes for Problems Involving Fluid Flow," Comp. Meth. Appl. Mech. Eng., Vol. 9, pp. 151-162.
- Reynolds, W. C. (1976), "Phenomenological Turbulence Modeling," in Ann. Revs. Fluid Mech., Vol. 8, p. 183.
- Roache, P. J. (1982), Computational Fluid Dynamics, Hermosa Publishers, Albuquerque, N.M.
- Rodi, W. (1980), Turbulence Models and Their Applications in Hydraulics, Intl. Assn. for Hydraulics Research, Delft.

- Rodrigue, G., and J. Simon (1983), "A Generalization of the Numerical Schwarz Algorithm," Lawrence Livermore Laboratory Report #UCRL-90030.
- Saltzman, J., and J. Brackbill (1982), "Applications and Generalizations of Variational Methods for Generating Adaptive Meshes," in Numerical Grid Generation (J.F. Thompson, ed.), North-Holland, New York.
- Shyy, W. (1985), "A Study of Finite Difference Approximations to Steady-State, Convection-Dominated Flow Problems," Jnl. Comp. Phy., Vol. 57, pp. 415-438.
- Smith, G. D. (1978), Numerical Solution of Partial Differential Equations: Finite Difference Methods (second ed.), Oxford University Press, London.
- Spalding, D. B. (1972), "A Novel Finite-Difference Formulation for Differential Expressions Involving Both First and Second Derivatives," Int. J. Num. Methods Eng., Vol. 4, p. 551.
- Starius, G. (1977), "Composite Mesh Difference Methods for Elliptic Boundary Value Problems," Numer. Math., Vol. 28, pp. 243-258.
- Stetter, H. J. (1978), "The Defect Correction Principle and Discretization Methods," Numer. Math., Vol. 29, pp. 425-443.
- Stoutemyer, D. R. (1972), "Numerical Implementation of the Schwarz Alternating Procedure," Ph.D. Thesis, Dept. of Computer Science, Stanford University, Calif.
- Tang, W. P., W. Skamarock, and J. Olinger (1985). To appear.
- Thompson, J. F. (1984), "Grid Generation Techniques in Computational Fluid Dynamics," AIAA Jnl., Vol. 22, No. 11, pp. 1505-1523.
- Thompson, J. F. (1983), "Dynamically-Adaptive Grids in the Numerical Solution of Partial Differential Equations," IFIP Conference on PDE Software, Soderkoping, Sweden.
- Van Leer, B. (1979), "Towards the Ultimate Conservative Difference Scheme. V. A Second-Order Sequel to Godunov's Method," Jnl. Comp. Phys., Vol. 32, pp. 101-136.
- Zabusky, N. J., and G. S. Deem (1971), "Dynamical Evolution of Two-dimensional Unstable Shear Flows," Jnl. Fluid Mech., Vol. 47, Pt. 2, pp. 353-379.
- Zebib, A., and G. M. Homsy (1984), "Laminar Flow Over a Backward-Facing Step," Manuscript #84-07, Center for Large Scale Scientific Computation, Stanford University, Stanford, CA.
- Zebib, A. (1984), personal communication.

Appendix A

QUANTIFICATION OF THE STRENGTH OF THE VISCOUS-INVISCID INTERACTION

In this appendix, we show how the strength of the viscous-inviscid interaction can be quantified for the flow in the inlet region of a plane channel. The same methodology can be applied to other flows. It is required only that the boundary-layer displacement thickness can be related to velocity in the inviscid, outer region.

The displacement thickness δ^* for a constant-density fluid is defined as:

$$\delta^* = \frac{1}{V_o} \int_0^{\delta} (V_o - v) dy \quad (\text{A.1})$$

where δ is the boundary-layer thickness, V_o the velocity outside, and v the velocity inside the boundary layer (see Fig. 3.1). δ^* is a measure of the boundary-layer thickness. For the flow over a flat plate, $\delta^* \sim \delta/3$.

The displacement thickness has the following significance. The uniform, inviscid flow at velocity V_o through a channel, whose height L has been reduced (displaced) by an amount equal to δ^* on each end, will have the same volumetric flow as the viscous flow through a channel of height L . The inviscid flow is coupled to the viscous flow in the boundary layer through this displacement effect.

The displacement thickness is related to V_o by:

$$V_o = \frac{Q}{L - 2\delta^*} \quad (\text{A.2})$$

where Q , the volumetric flow through the channel, is:

$$Q = \bar{V}L \quad (\text{A.3})$$

and \bar{V} is the average velocity in the channel. Substituting (A.3) into (A.2) gives:

$$V_o = \frac{\bar{V}}{1 - 2\delta^*/L} \quad (\text{A.4})$$

where

$$V'_0 = \frac{V_0}{V} ; \quad \delta^{*'} = \frac{\delta^*}{L} \quad (A.5)$$

If the displacement thickness is increased by an amount $\Delta\delta^*$, the outer velocity becomes:

$$V_0(\delta^* + \Delta\delta^*) = \frac{1}{1 - 2(\delta^* + \Delta\delta^*)} \quad (A.6)$$

where we have dropped the " ' ". Subtracting (A.4) from (A.6) and dividing the result by (A.4) gives the following relationship for the relative change in V :

$$\frac{\Delta V_0}{V_0} = \frac{V_0(\delta^* + \Delta\delta^*) - V_0(\delta^*)}{V_0(\delta^*)} = \frac{1 - 2\delta^*}{1 - 2(\delta^* + \Delta\delta^*)} - 1 \quad (A.7)$$

Let the change in δ^* be:

$$\Delta\delta^* = (K-1) \delta^* \quad (A.8)$$

where K is a sensitivity parameter. Substituting (A.8) into (A.7) gives the desired result:

$$\frac{\Delta V_0}{V_0} = \frac{2(K-1) \delta^*}{1 - 2K\delta^*} \quad (A.9)$$

Equation (A.9) quantifies the strength of the coupling between the boundary layer and the outer, inviscid flow.

$\Delta V_0/V_0$ is plotted in Fig. 3.2, for $K = 2, 10$. On a coarse grid, the boundary layer may be smeared to twice its actual size, $K = 2$. The relative change in V_0 should be no greater than the maximum allowable error. For $K = 2$ and maximum allowed error, 0.1%, the coupling is weak for $\delta^*/L < 10^{-3}$, and strong at larger δ^*/L .

END

DT/C

8-86